COMPUTERVISION

CADDS 4

BASIC

OPERATORS

COURSE

VERBS   NOUNS   MODIFIERS

DYNAMIC MENUS

DEMO | DATA INPUT | DIM

BLANK	DIV	REV	UNBL	ARC	FIL	OFFS	HOR	RAD	LNG	STPT	MODEL	AX	USER DEFINABLE KEYS
CHA	ECHO	ROT	VER	CHAM	GRID	POI	VER	DIAM	A B	ENDPT	DRAW	AY	
CONST	EDIT	SEL	INS	CIR	GROUP	TEXT	PAR	AGO	-TRIM	INCL	CPLANE	AZ	
COPY	MEAS	STR		CPL	LAYER	VIEW	PERP	AEND	DTRIM	EXCL	DPLANE	ILNG	
DEF	MIR	TRAN	DEL	DIST	LEN		TANTO	ANG	COPY	FROM	ON	NDIV	
DISS	PRO	TRIM		ENT	LIN		INTOF	THRENT	SAMLAY	TO	OFF		EDIT KEY SAVE KEY LIST KEY

DEL C DEF FROM
LIST P SEL
ECHO ON L ECHO OFF

| SET | DG | ? ! | SPACE | REPA | FILE PART | ↑E US DEL | : | - | " | . | : |

Numeric / calculator block

7	8	9
4	5	6
1	2	3
0	.

XO YO ZO

X	Y	Z
IX	IY	IZ
A	R	D
N	IA	IR	DEND

| · |

S H O W	+	−
	×	÷
(	)
SPACE
:

ZOOM DRAW
IN   OUT ALL) WIN:
R   A

SCROLL DRAW
← → ↑ ↓
R   D

SAVE
REST
LIST
DEL

I M A G E

ZOOM VIEW
IN   OUT
R   A
ALL: WIN:

SCROLL VIEW
← →
↑ ↓
R   P

DYN VIEW
SET VIEW
RESET VIEW
NAME ALL)

:

Fraction / decimal conversion table

1/64	.015625
1/32	.03125
3/64	.046875
1/16	.0625
5/64	.078125
3/32	.09375
7/64	.109375
1/8	.125
9/64	.140625
5/32	.15625
11/64	.171875
3/16	.1875
13/64	.203125
7/32	.21875
15/64	.234375
1/4	.250
17/64	.265625
9/32	.28125
19/64	.296875
5/16	.3125
21/64	.328125
11/32	.34375
23/64	.359375
3/8	.375
25/64	.390625
13/32	.40625
27/64	.421875
7/16	.4375
29/64	.453125
15/32	.46875
31/64	.484375
1/2	.500

33/64	.515625
17/32	.53125
35/64	.546875
9/16	.5625
37/64	.578125
19/32	.59375
39/64	.609375
5/8	.625
41/64	.640625
21/32	.65625
43/64	.671875
11/16	.6875
45/64	.703125
23/32	.71875
47/64	.734375
3/4	.750
49/64	.765625
25/32	.78125
51/64	.796875
13/16	.8125
53/64	.828125
27/32	.84375
55/64	.859375
7/8	.875
57/64	.890625
29/32	.90625
59/64	.921875
15/16	.9375
61/64	.953125
31/32	.96875
63/64	.984375
1	1.000

GETDATA

WIN	PWIN	VWIN	END	REF	LIN	POI
IN	OUT	CROSS	ORG	C	CIR	ARC
⌐REPA.	CHN	INTOF	-C	FIL	ENT

GRO LOC VERT

DRAW   VIEW

| ZOOM SAVE | ZOOM ALL | LAY | DYNV⌐ | ZOOM VIEW |
| SCROLL RESTORE | | CPL NAME | RESET VIEW | SCROLL VIEW |

| ACT PAR | ACT DRA | SIZE | A |
| | | B D | E |

| ECH FRA OFF | LIST DRA STAT | HGT WDT UNITS |

| DEF VIE | CPL DIST PRP DMIN | LIST VIE |
| | FOLD FACE PRL DMAX | DEL VIE |

| SEL MODE MODEL | PLOT HARD | DYNAMIC ON | NAME ALL | REV VIE CLIP |
| SEL MODE DRAW | DUMP STEXT | DYNAMIC OFF | SCALE : | REV VIE LOC |

| COUNT ENT | PACK DBA | CHECK DBA | CHA EXT LEAST | LIST EXT |
| EXIT F | EXIT CADDS | OS | RESU SYS | EXIT Q |

KEYFILE.TRMD.MENU

DAILY SCHEDULE

COMPUTERVISION
CADDS4 BASIC OPERATORS COURSE

GENERAL BACKGROUND INFORMATION

The following paragraphs present practical infor-
mation to the operator on the use of CADDS 4.
These paragraphs explain:

* Command Format

* Command Punctuation

* Control Punctuation

* Login

* Logout

* Crash Recovery

* Operations Levels

* Model Mode and Drawing Mode

* Parts, Models and Drawings

* Part Names

* Coordinate Systems

* On Line Documantation

* Data Input Prompts

* Getdata

**COMMAND
FORMAT**

The CADDS graphics system is designed for easy use by the operator. It is broken down into several parts that allow for a number of different combinations of command types. The basic format is:

<u>VERB</u>	<u>NOUN</u>	<u>MODIFIERS</u>		<u>GETDATA</u>
INS	LIN	VER	:	X0Y0,X1Y1
	WHAT			WHERE
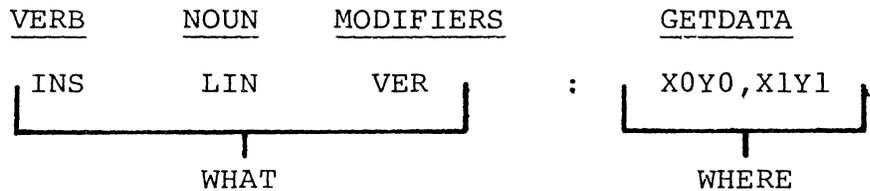
VERB-Every command will have a verb. Some commands will consist of only a verb.

NOUN-A command may or may not have a noun. No command will have more than one noun.

MODIFIERS-A command may or may not have modifiers. A command may have more than one modifier.

COLON-Once the colon is entered, the data input software takes control.

Generally speaking, the verb, noun, and modifiers in front of the colon tell the system what function is to be performed, and the data after the colon tells the system on which entities, or where, the function is to be performed.

GETDATA-Processed by the data input software. Consists of coordinate or entity information.

The command format in this manual is presented in two forms. Upper and lower case letters and numerals are entered by the operator. COMPUTER TYPE indicates system prompts or responses.

For example:

    INS LIN HOR LNG3:MODEL LOC

    Entered by operaotr   System prompt

To distinguish between system and user responses
in the dialogue examples contained within this
manual, different type syles are used.

    This Type - Indicates user input

    THIS TYPE - Indicates system response or
                  output.

Other conventions are indicated below.

    (CR) - Indicates a carriage return.

    ( ) - Indicates the information within
          the brackets is optional.

COMMAND PUNCTUATION

CADDS 4 command punctuation is standardized; punctuation marks retain the same meaning throughout all commands. The following table lists and describes the various command punctuations.

CADDS 4 COMMAND PUNCTUATION

Punctuation	Use
Blank	Used optionally between keywords.
, (comma)	Used between explicit coordinates, items in a list, and groups of numbers.
; (semicolon)	Used between clauses, as in switching modes from identification to digitizing.
: (colon)	Used to enter the coordinate input entity input model, or to reenter the same command without reissuing the verb/ noun combination. When used to reissue the command, the colon terminates the input, executes the command and then reenters the same command for new coordinate input, thus preserving all modifier selections from the previous command specification.
. (period)	Used to terminate the current command input, execute the command, and then reenter the same command without reissuing the verb/noun combination. When a period is entered, the command reenters the modifier process and awaits further user direction.
(CR) (carriage return)	Used to terminate the current input and execute the command.

**CONTROL
PUNCTUATION**

CADDS 4 control punctuation is standardized;
punctuation marks retain the same meaning
througjout all commands. The following table
lists and describes tha various command
punctuations.

CADDS 4 CONTROL PUNCTUATION

Punctuation	Use
CTRL E	Aborts the command. Control returns to the operator at the CADDS system level. The display and part database are both returned to the previous state.
CTRL B	Stops processing and the operator enters one the following replies:
	Q - Halt program execution and return to CADDS level prompt.
	1 - Process the command through one process loop until this breakpoint is reached again or command terminates.
	Any other character causes the command to resume execution.
	The display and part data base remain in the state they were in prior to the halting of the command.
CTRL Q	Aborts processing and returns control to CADDS level prompt.

LOGIN	An operator must log in to the computer in order to use the system. Logging in is the users way of informing the computer that they are ready to begin work.
INITIAL LOGIN	The first operator to log onto the system must type in the date and time. The computer will automatically request this information from the operator. The system automatically performs an FMCLEAR also.
CTRL-R n	To initiate the login proceedure the operator must type CTRL-R followed by n. CTRL is the control key and R is the letter key R. CTRL and R are pressed simultaneously. 'n' is the task number.
EXAMPLE	Operator inputs are preceeded by (O).

        (O) CTRL-R n

           **** CGOS 200/10 ****

           ** FMCLEAR **

           TYPE DATE AND TIME: MM-DD-YY,(HH:MM)

        (O) 12-4-82,16:27 (CR)

           TYPE NAME, NUMBER
        (O) GUEST (CR)
           TYPE PASSWORD
        (O) (CR)

           ** TASK n INITIATED **
           n>SELECT BATCHMAX=0,LP=OFF
           n>

SYSTEM PROMPT	On the Computervision System the prompt at operating system level is n

'n' is the task number the operator has logged onto.

LOGOUT

The operator must logout whenever a work session is completed to let the computer know that work has ceased and that a particular account should no longer be charged for computer time.

1. A logout may only be performed when the task is at systems level.

PROCEDURE

(O) n>LOGOUT (CR)

    MINS ELAPSED: 48 COST: $6.23
    ** TASK n TERMINATED **

(O) CTRL-R N   N is the task number.

     TYPE NAME, NUMBER

(O) GUEST (CR)

     PASSWORD

(O) (CR)

     SYSTEMS LOGIN MESSAGE

(O) X TRG (CR)

     SYSTEMS MESSAGE

(O) CADDS (CR)

     SYSTEMS MESSAGE

(O) #N# ACT PAR TRG.XXX.EX.1

     ENTERING NEW OR OLD PART

(O) #N# ACT DRA D1

When work has been completed:

(O) #N# EXIT PAR Q OS
         -OR-
    #N# EXIT PAR F (FILENAME (OPTIONAL)) OS

(O) LOGOUT

CRASH RECOVERY PROCEEDURES

The process outlined below describes how to recover a part
when a system crash occurs.

--CRASH--At this point the system will no longer respond
         to operator input. The menu will be inactivated,
         the keyboard will not respond and the terminal
         will be logged out. No logout message will
         appear on the screen.

         To begin work again the system must be booted up
         again. After this all tasks must be logged in
         again. All operator inputs are identified in the
         following example by the symbol (O). Explanatory
         notes are identified by an asterisk(*).

(O) CTRL-R N*   *Be certain to use the same
     TYPE NAME, NUMBER   task number you were logged
(O) GUEST (CR)   in under prior to the crash
     TYPE PASSWORD
(O) (CR)

     ** TASK N INITIATED **

SYSNEWS, BCD,LCLNEWS
1- 1-91   0:01:08

****
***   CADDS 4   REV. 2-01-A   ***
****

(O) N>X TRG
     N>ENABDIR TRG=C, SYSCATLG=L

     DIRECTORY ENABLED

     N>OK
     ??
(O) N>CADDS
     CADDS 4 REV. # 2-01-A   2-01-82   16:15:48

     TYPE OK TO CRASH RECOVER
(O) OK*   *Typing OK here is the first
     INPUT DEVICE IS SD   step in recovering a part
                                   assuming it was not lost in
                                   the system when the crash occured.

```
       TYPE OK TO REGENERATE GRAPHICS
(O)  OK*                            *Typing OK here places the
                                    information about the part
                                    in a temporary file and does
                                    NOT display the recovered
                                    part on the screen.

     PART WAS SUCCESSFULLY RECOVERED AND ACTIVATED UNDER A
     TEMPORARY NAME. FIGURE FILES WERE NOT RECOVERED. BE SURE
     TO FILE UNDER NEW NAME.

(O)  #N#FILE PART (filename)*       *The part, although unseen, must
                                    be filed under a name that can
(O)  #N#EXIT PAR Q*                 be identified.

(O)  #N#ACT PAR (filename)          *Even though the temporary file
                                    name is not known it must be
     ENTERING OLD PART              exited in order to continue.

(O)  #N# ACT DRA D1*                *This completes the process and
                                    allows work to continue at the
                                    point just before the crash.
                                    There is always the possibility
                                    that a part will not be re-
                                    covered and, thus, reinforces
                                    the importance of filing often
                                    to avoid losing a great deal of
                                    time and work on a part.
```

Drawings are the means by which the model is
presented and detailed. A drawing contains
one or more independent pictures (views) of
the model. Each view contained within a draw-
ing may be tailored to express some particular
quality of the model. Each drawing may be
worked on independently allowing the user to
create as many drawings as are desired.

Since the purpose of a drawing is to present
and detail the model for a specific purpose,
certain changes to the appearence of the
model may be desired. To accomplish this,
CADDS 4 allows the user to edit the model
within a drawing without changing the database
description of the model. Drawings use a copy
of the database description of the model,
eliminating the need to recreate the model for
each drawing. Drawings are dependent upon the
database description of the model in that all
revisions and changes made to the model (in
model mode) are reflected, where appropriate,
in all drawings.

PART NAMES

Every drawing, layout, sketch, etc. that you want to create on the CRT must be given a name before you can start.

The part name can be made up of several parts separated by periods.

XXX.DDD.LLL

RESTRICTIONS

1. 20 characters maximum between periods.

2. Total number of characters: including periods cannot exceed 60 characters.

3. Cannot have embedded blanks in the part name.

Naming convention for training parts:

TRG.DSB.EX.1

- Part Number
- Operators Initials
- Training
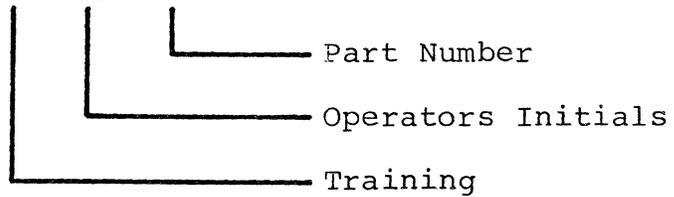
OPERATIONS LEVELS

There are two levels at which the system is operated:

    1. The systems level

    2. The CADDS level

You can tell which level you are in by the prompt

    1. n> for systems level

    2. #n# for CADDS level

The CADDS level is where the drawings and the layouts are created.

The systems level is for reading and writing mag tapes, fmclear, login, logout, and other non graphic type operations.

MODEL MODE AND DRAWING MODE

Two operational modes exist within CADDS 4 which distinguish between operations of drafting and designing natures. These modes are referred to as model mode and drawing mode. Operations performed on the model within drawing mode affect the presentation of the -odel within that drawing only (i.e., the batabase description of the model is not affected). Commands which may be used only within a specific mode are said to be "mode-dependent".

PARTS, MODELS, AND DRAWINGS

A CADDS 4 part consists of a database description of the model and any number of drawings necessary to describe the model. The model and all drawings pertaining to that model are stored under a part name. The user accesses the model and drawing by using the part name and specifying the desired drawing.

A model is the database description of the abstract object (or product) which is being designed or produced. All geometry and data required to define the model is stored automatically by the system. Models may be two- or three-dimensional. Certain graphical entities (generally 3-D) may be used only by the model.

COORDINATE SYSTEMS

Coordinate systems, views, and construction planes are used to provide a means by which the model may be viewed and input entered where desired. Two distinct special coordinate systems are used by CADDS 4. These are:

1. Model Space

2. Drawing Space

MODEL SPACE

Model space is the coordinate system in which the model resides. Model space is an infinite, universal space which is also referred to as "real world space". The model always resides in model space; representations (views) of the model are created in local coordinate spaces (construction space). The model may always be thought of as aligned with the specific coordinate axis associated with model space. Conventionally, the modal space coordinate axis is represented as follows (relative to the display); positive X, to the right; positive Y, up; and positive Z, out from the screen.

DRAWING SPACE

Drawing space is a two-dimensional coordinate system which defines the frame of reference for a drawing.

VIEWS AND CONSTRUCTION PLANES

Views and construction planes are used to define a local coordinate space in which the model may be viewed in different ways (from different angles, position, etc.). A construction plane is a pre-defined or user-defined plane on which digitizes are projected (if the construction plane is activated). Construction planes also define the view orientation as different from model space, allowing the model to be displayed from different angles, and the digitizes to be projected to the activated construction plane. Construction planes may be activated in a pre-defined view to allow the user to project digitizes upon a plane other than the currently activated construction plane.

Views are the means by which the model, or part of the model, is scaled and positioned within the user-defined view clipping boundaries. This model is placed within the view relative to the user-defined view origin and the view

orientation (defined by the associated view construction plane). Views establish a view coordinate system which is used for all explicit coordinate input when the associated view construction plane is activated.

Seven pre-defined construction planes exist which may be used by the operator. If a construction plane is not activated by the user, the system defaults to the pre-defined top construction plane (regardless of whether or not the user has used this construction plane to define a view). The top construction plane exists in model space, and may be thought of as the system's representation of model space.

DATABASE UNITS

Every model, or drawing of a model, can be considered to exist in a space of specific dimension. Every dimension may be expressed in terms of a unit of measure and a magnitude (how many units). CADDS 4 allows the user to explicitly select the dimensional unit in which the drawing or model will exist from a list of system supported units. These dimensional units are referred to as database units. CADDS 4 supports the following database units: inch, foot, mile, millimeter, centimeter, meter, and kilometer. Any of these units may be modified by the use of a multiplication factor such as 3 x foot, 0.001 x inch, etc. After specification of a multiplication factor, all drawing or model size information (magnitudes) are interpreted by the system as some multiple of the selected unit (i.e., dimensional unit X multiplication factor).

Two types of database units are supported by CADDS 4: model units and drawing units. Model units express the dimensions of the model in its dimensional space, likewise, drawing units express the dimensions of the drawing. One model unit may be selected per CADDS 4 part, and one drawing unit per drawing.

ON-LINE DOCUMENTATION

You can obtain an on-line description of any valid CADDS 4 command. They are activated by the use of the special characters:

    ? (question mark)
    ! (exclamation point)

LISTING OF VERBS AND NOUNS

The ? is used to list verbs, associated nouns and/or associated modifiers.

    verb?   - Lists all nouns used with a verb.

    verb noun? - Lists all modifiers available for this command.

    characters? - If a few characters are entered, but not enough to identify a verb, a ? will list all verbs begining with those characters.

COMMAND DOCUMENTATION

The ! is used to produce a description of a particular command.

    !verb noun - Lists a description of that verb/noun combination.

TO GET OUT

(ESCAPE)

(CNTR Q)

LIST PART CATALOG, A---)
    "   "   "   TRG.EB.)

DATA INPUT
PROMPTS

Data input takes control after the colon is entered. Data input is a general purpose means of inputting data, whether it be explicit data or graphic data input through digitizes.

The standard Data Input prompts include the selected mode (Model/Draw) as the first part of the prompt in upper case, along with the intent of the input (entity retrieval, co-ordinate specification) in lower case as the second part.

These parts are separated by a blank, and a blank is appended to the total prompt.

Part 1: Mode prompt

    a. MODEL   - All digitizes refer to model space.

    b. DRAW   - All digitizes refer to drawing space.

    c. DRAW/MODEL - Digitizes may pick up either model or drawing entities.

Part 2: Input intent

    a. ent   - Identifies entities

    b. loc   - Identifies a coordinate locstion

EXAMPLES

INS LIN:MODEL LOC

EDIT TEXT:DRAW ENT

GETDATA

COORDINATE
LOCATIONS

There are two ways to identify input locations to the system.

1. The pen is used to point to or digitize the location.

2. Explicit input - the coordinates are typed in. Two types of coordinates can be used.

    A. Cartesian Coordinates

        Absolute coordinates in the form of XnYnZn. This is measured from X0Y0Z0.

        Incremental coordinates in the form of IXnIYnIZn. This is measured form the previous coordinate. For example:

            IX3 means - Starting at the last coordinate entered, move 3 inches in the positive X direction.

    NOTE: A comma is used to separate coordinates X2Y2,X5Y4.6

    B. Polar Coordinates

        Absolute coordinates - RnAnZn
        Incremental coordinates - IRnIAnIZn

        A is measured counter-clockwise from the horizontal.

        R is radial from (0,0).

To iterate (repeat) a coordinate, that has been input using explicit coordinate input, a number of times, end the input with an 'N' followed by an integer.

                INS POI:MODEL LOC X0Y0,IX1IY1N5

MODIFIER
MODES

Data can be located by referencing other entities - lines, arcs, circles, etc. The pen and tablet are used to point to or digitize an entity. Watch for the attention marker to appear on the entity identified to be sure the right entity was identified. (The attention

marker is a small square that appears on the
origin of the entity on the storage tube
terminals and a small flashinf 'X' that appears
on the entity near the digitize on the Insta-
View terminals.

Endpoint Mode - END

> To refer to the endpoint of a line or
> other curve. The endpoint of an entity
> is specified by digitizing the entity
> near the endpoint desired.

Origin Mode - ORG

> To refer to the origin of an entity, for
> example, the origin of a circle is the
> center and the origin of a line is the
> midpoint.

Refernce Point - REF

INS LIN /ER LEN H; MOD .LOC ORG REF d, LOC IY-3

> The ref data modifier can be used anywhere
> that coordinate information is legal in
> Getdata. It causes the following location
> specification to be used as a reference
> only, not as a data input to the command.
>
> The location specification which follows
> REF can take any of the previously
> described modifiers.

Local Center - C

> Moves the origin for the remainder of the
> current command. It requires a location
> input similar to REF.

Near

> The naer modifier selects the nearest
> point on the specified entity.

Intersection Of - INTOF

> The INTOF modifier is used to establish a
> location at the geometric intersection of
> two selected entities or of one entity
> (specified first) with several others.

NOTE: You remain in the chosen mode until
      another modifier is used which specifies
      a different mode or until Getdata is
      exited.

**ENTITY
SELECTION**

The Getdata modifiers PNT, LIN, ARC, etc. may
be used to restrict the choice of entities.

Also, the modifier WIN (window) or PWIN
(polygon window) can be used to identify
more than one entity at a time.

EXAMPLES

```
                              P1   P2   P3   P4   P5   P6
#n#INS LIN:MODEL LOCX1Y0,X3Y3,X5Y4,Y5,X1Y7,X5 (CR)
```
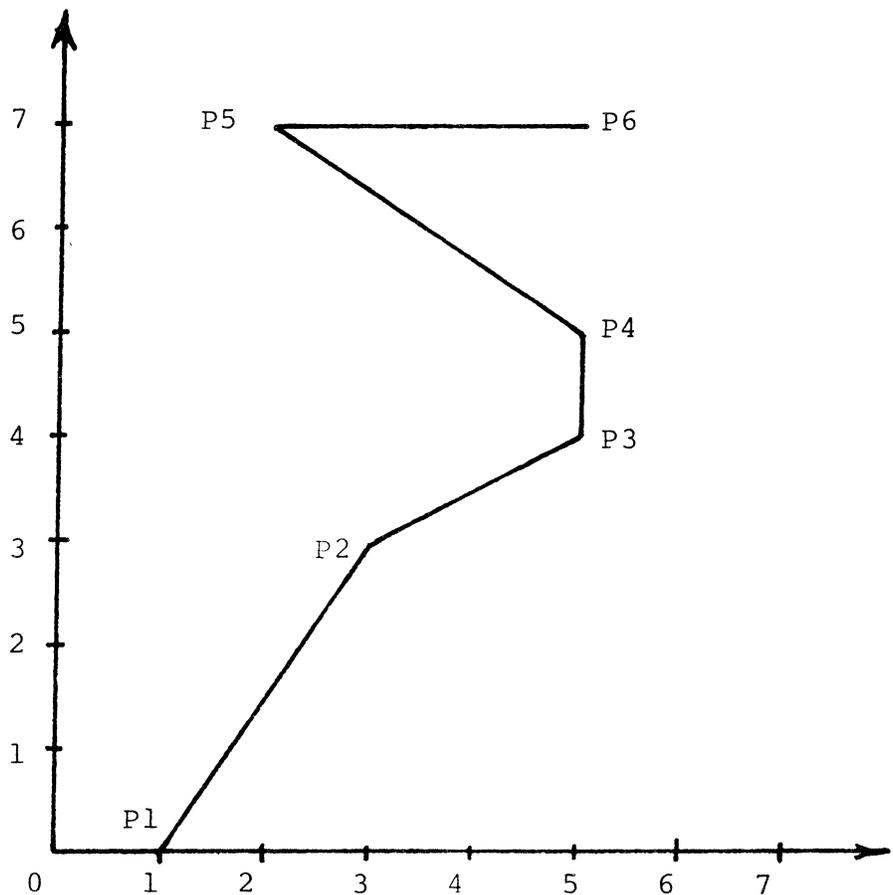
EXAMPLES

#n#INS LIN:MODEL LOC d1d2d3;d4d5d6d7d8
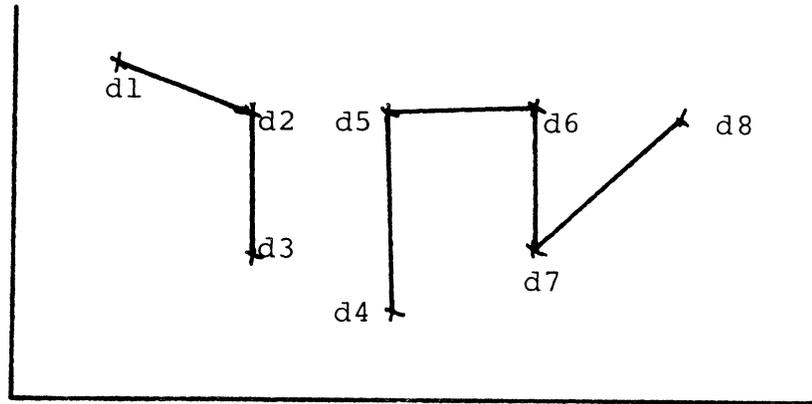
                                    P1   P2   P3   P4
#n#INS LIN:MODEL LOC X0Y0,R2A45,R3,A60
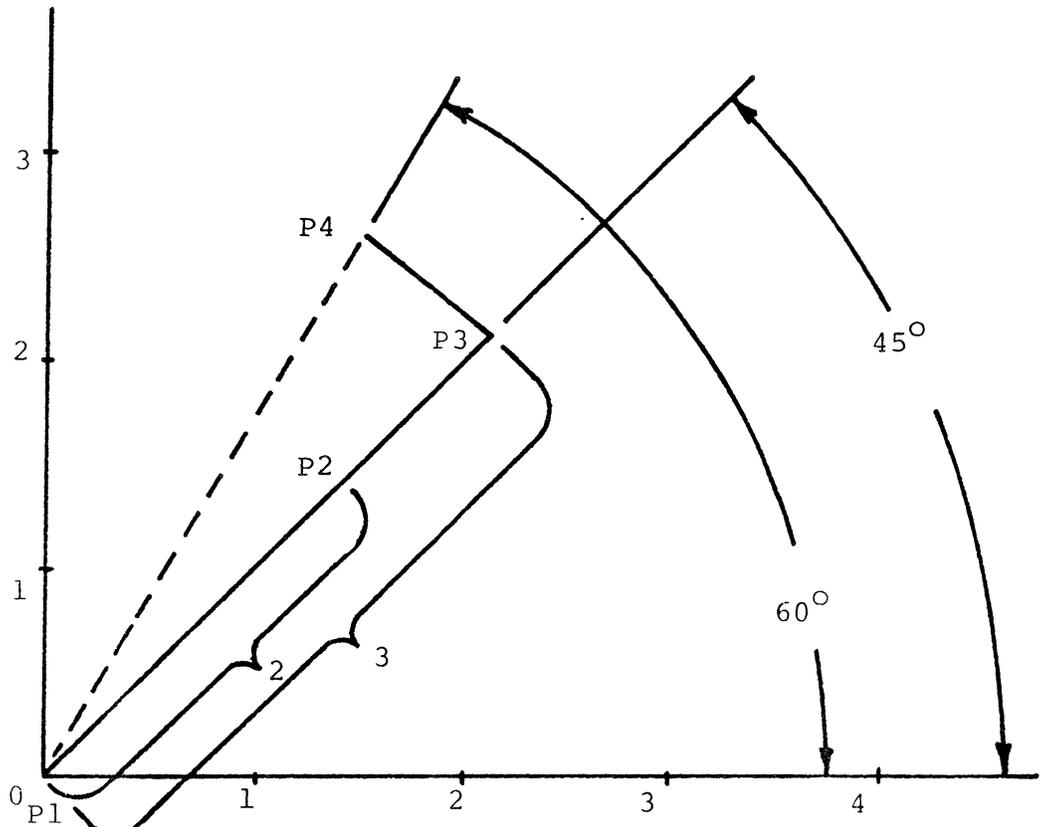
P1   P2-P6

#n#INS LIN:MODEL LOC X2,Y0,IA60N6

P3

P2

60°

P1

2 in.

P4

P5

P6
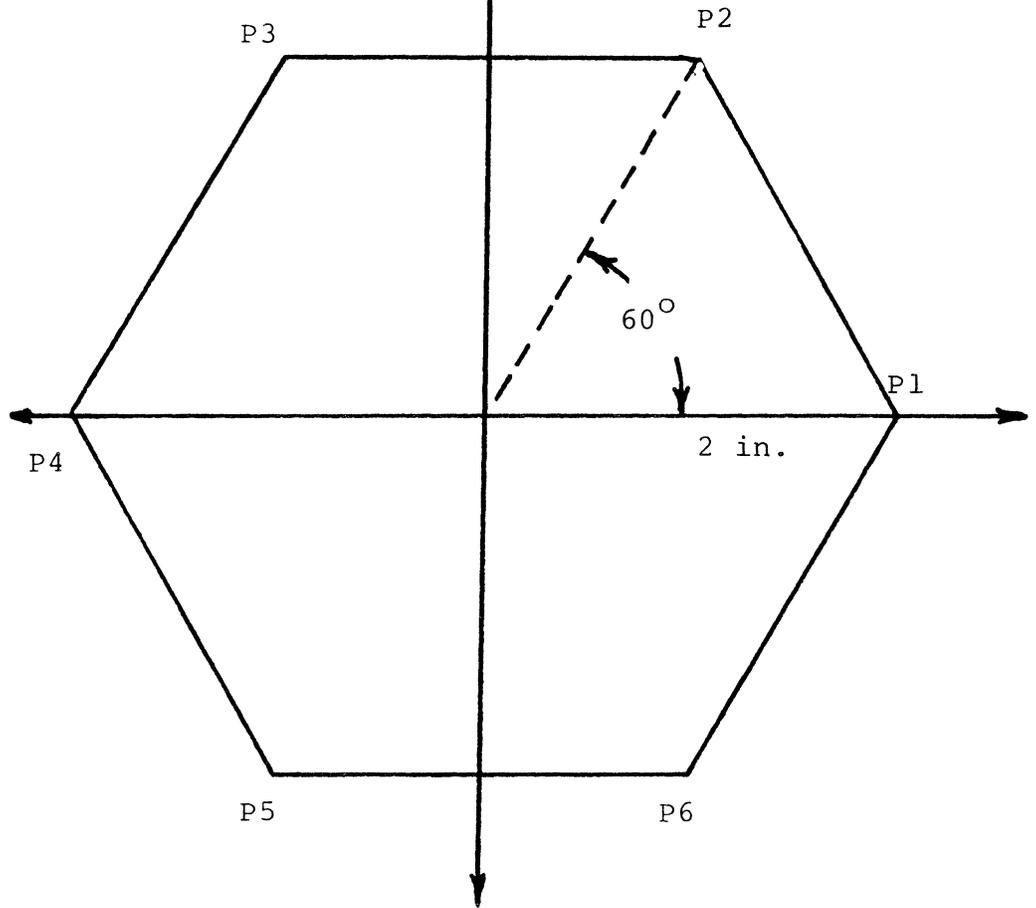
#n#INS CIR RAD1:MODEL LOC REF ENDd1LOCIX3

3.000

*d1
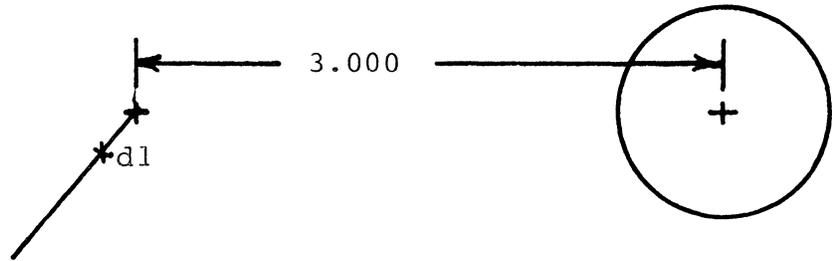
#n#INS LIN:MODEL LOC X15Y15,NEARd1

X15Y15

X
d1
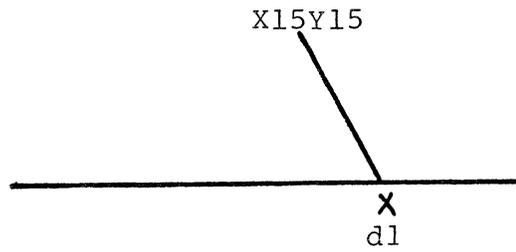
#n#INS CIR RAD2:MODEL LOC INTOFd1d2

X d1

X d2

KEYFILE TRG4

OUT	IN	VERB				NOUN				MODIFIERS							PUNCTUATION										DISPLAY CONTROL		ANALYSIS	

OUT	IN		VERB				NOUN				MODIFIERS						PUNCTUATION											INS	DISPLAY CONTROL		ANALYSIS
EXIT PART Q	ACT PART	ERS	DEL	ECH	INS	LIN		LAY		HOR	VER	ANG	PERP	PARA	TANTO	LNG	:•	SPACE	+	0	−	X	IX	AX	INS TEXT ▼▼	INS LDIM	SAV	IMAG	MEAS	LEN
EXIT PART F	ACT DRAW	BLA	UNB	CHA	SEL	POI		ENT	ALL	ON	INTOF						•	▼▼	1	2	3	Y	IY	AY	▼▼	INS ADIM	RST	ZOOM VIEW	VERIF ENT	ANG
OS	DEF VIEW TOP	ZOOM	SCRO	DYN	RESET	ARC	CIR		VIEW	RAD	DIAM	AGO	AEND				,	?	4	5	6	Z	IZ	AZ	FONT	INS RDIM	ZOOM DRAW	WIN		DIST
EXIT CADDS	ECH FRA	ROT	MIR	TRAN	COPY	FIL	CHAM		DRAW	LNGA	LNGB			INCL	EXCL		;	!	7	8	9	A	B	C	HGT	INS DDIM	IN	OUT		AREA
RESUME SYS	FILE PART	VERIF FNT	PROJ	TRIM	EDIT	KEY	TEXT	PART	GRID	ON	OFF	IN	OUT	DG			D	E	F	G	H	I	J	K	WDT		SCROL DRAW	LEFT		
	SEL MODE DRAW	SAV	COUNT	LIST	PLOT	LAB	XHA	HARD	IMAG			END	REF	WIN	NEAR		L	M	N	O	P	Q	R	S	ANG		UP	RIGHT		
	SEL MODE MODL	RST		STRETCH	DEF	BSPL		CPL	↙	↑E	↑H	ORG	LOC	INTOF	ALL	SPACE	T	U	V	W	X	Y	Z	↙			DOWN	SCROL VIEW		
					REPA	ECH LAY A REPA																					DYNA VIEW	RESET VIEW		
					:•												:•										SET VIEW			
INS RSUR	INS BSUR	INS TCY	INS SREV	CUT SUR	CUT PLA	RSUR																								
MESH	LAY	LOWBN	LOWAN			BSUR											DEF CPL	AX	AY	AZ	ANG									
PATCH	BND	HBND	HIANG			TCY											SEL	CPL												
WEDGE	AMIN					SREV											ECH	CPL	NAME	OFF	L									
HEDGE	AMAX					CHA SGRA											DEF VIEW	TOP	FRONT	RIGHT	BOT	LEFT	BACK							
	AINT																ISO						DEL VIEW						
INIT MPROP	CALC MPROP	LIST MPROP																												
XSECT	PROJ	DENS	LAY	ILAY	NSTEP	STEP																								
ROT	HIANG	LOWAN	HIBND	LOWBN																										
																								YES	NO	OK				

SYSTEM CONTROL COMMANDS

System Control commands are used to activate and deactivate a CADDS 4 work station, to move between the Operating System environment and the CADDS 4 environment, to set up alternative verb/noun combinations for altering the message system, and to perform certain system utility functions.

The commands include:

      EXIT CADDS
      RESUME CADDS
      RESUME SYSTEMS

COMMAND	CADDS
DESCRIPTION	The CADDS command takes the user from the Operating Systems (OS) level to the CADDS 4 environment with a standard or user-defined set of default parameters.
SYNTAX	CADDS (CR)
EXAMPLE	#n#CADDS (CR)

COMMAND	EXIT CADDS
DESCRIPTION	The Exit Cadds command is used to exit the non-graphic stage of CADDS and return to the operating system.
SYNTAX	EXIT CADDS (CR)
ERROR MESSAGES	If the Exit Cadds command is used in the graphic stage of CADDS the message

PART ACTIVE, COMMAND IGNORED

will be displayed.

RESTRICTIONS	The Exit Cadds command can not be used while a part is active.

COMMAND	RESUME CADDS
DESCRIPTION	The resume CADDS command transfers control from the Operating Systems level to the CADDS 4 level. This command restores a CADDS session suspended by a Resume System command. Resume CADDS is the only way to enter CADDS after CADDS activity has been suspended.
SYNTAX	RESUME CADDS (CR)
WARNING	Do not break the execution of a Resume command.
ERROR MESSAGES	CADDS SUSPENDED

An attempt is made to enter CADDS while CADDS is suspended. Returns to the Operating System command level.

CADDS IS SUSPENDED, LOGOUT ABORTED

Logging out when CADDS is suspended. Returns to the Operating System command level.

CADDS NOT SUSPENDED

An attempt to RESUME CADDS when CADDS is not suspended. Returns to the Operating System command level.

COMMAND	RESUME SYSTEMS
DESCRIPTION	The Resume Systems command temporarily suspends CADDS operation and transfers control to the Operating System command level.
SYNTAX	RESume SYStems (CR)
WARNING	Do not break the execution of this command.
INTERNAL ERRORS	INTERNAL ERROR: REST OF MESSAGE
	The Resume Systems command detects an internal error and returns control to CADDS.

PART STRUCTURING COMMANDS

`

Part structuring commands are the first to be used when creating or entering a CADDS 4 part. They permit the user to establish the operational part structure through which the model is viewed and manipulated.

This includes:

* Setting up the part and its parameters.
* Activating drawings and determining their parameters.
* Defining views and construction planes.
* Filing the part.

These commands have no effect on the model itself but rather on how the model is displayed.

These commands include:

ACTIVATE DRAWING
ACTIVATE PART
CONSTRUCT PART
DEFINE CPL (CONSTRUCTION PLANE)
DEFINE VIEW
EXIT PART
FILE PART

COMMAND	ACTIVATE DRAWING
DESCRIPTION	The Activate Drawing command is used to file the currently active drawing and to activate a different one, which may be either old or new. If old, it is displayed zoomed to the extents of the screen. If new, its size parameters are established either explicitly by command modifiers or from default values. A new or old drawing may be activated in either Model or Drawing mode.
	If drawing mode is in effect prior to activation, and a part is entered, the system automatically activates a drawing named DEFAULT, using the currently defined scale factor. However, drawing scale may be changed each time a drawing is activated.
SYNTAX	ACTivate DRAwing name (modifiers) (CR)
MODIFIERS	
oldname newname	(Required) Indicates a user-defined name for the drawing; limited to 15 characters.
HGTn WDTn	Specifies explicitly the height and width of the drawing when plotted true size. The drawing size effectively defines a final clipping boundary through which any view pictures and/or drawing graphics which try to extend further will be clipped. 'n' may be an integer or real number. Use these modifiers to achieve a vertical or horizontal format.
SIZE'...'	Allows input of one of several standard drawing size codes.

```
              A  - 8.5 x   11 in
              B  -  11 x   17 in
              C  -  17 x   22 in
              D  -  22 x   34 in
              E  -  34 x   44 in
             AO  - 841 x 1189mm
             Al  - 594 x  841mm
             A2  - 420 x  594mm
             A3  - 297 x  420mm
             A4  - 210 x  297mm
```

With this modifier, drawings are always oriented eith the longer dimension horizontal.

COMMAND	ACTIVATE PART
DESCRIPTION	The Activate Part command provides admittance into the CADDS 4 part environment. The characteristics for the new part and view planes are determined from the default parameter file used at CADDS activation time or from modifiers selected while in the CADDS non-graphical state. The mode (mode model) is determined by the user selected name, number, and password. If drawing mode is in effect prior to activation, a system-generated drawing named 'DEFAULT' is automatically activated as the part is activated. The user may then immediately begin drawing. Furthermore, if a drawing scale factor is in effect prior to part activation, then the DEFAULT drawing is automatically activated at the current scale factor.
SYNTAX	ACTivate PARt name (modifiers) (CR)
MODIFIERS	
oldname newname	Nmae of activated part.
EXAMPLE	ACT PAR TRG.XXX.LINES.EX.1 The new part,TRG.XXX.LINES.EX.1, is activated.
RESTRICTIONS	1. Part name may not exceed 60 characters, inclusive of "." used for catalog levels. 2. Drawing name may not exceed 15 characters. 3. No part name can begin or end with a "." or "&". 4. A part name currently in an active part cannot be activated. 5. The active part must be explicitly exited using the EXIT PART command.
ERROR MESSAGES	BAD PART NAME Output if partname or drawname exceeds their specified maximum number of characters. ALREADY AT PART LEVEL Output when user tries to activate a part presently in an active part.

COMMAND	CONSTRUCT PART
DESCRIPTION	The Construction Part command allows all, or a subset of the entities in the currently active part to be extracted and filed as another part.
SYNTAX	CONstruc PARt name (modifiers) (CR)
MODIFIERS	
MODEL	Select only model entities. Default to currently selected mode.
DRAW	Select only drawing entities.
ORG	During part construction a new origin. or base point for the constructed part may be specified. The three dimensional base point (X,Y,Z) so specified will become the origin point (0,0,0) of the constructed part. If no base point is specified, the origin of the active part, i.e., it's (0,0,0) point, will become the origin of the constructed part.
-ORG	Specifies that the origin of the active part is 0,0,0.
RESTRICTIONS	1. The limit of the total number of entities which may be extracted is 28,000. Entities digitized after this limit is reached are ignored.
	2. The constructed part may not be filed under the same name as the active part.
EXAMPLE	The following command will construct a part called VBLOCK consisting of only those entities that have been digitized. The new part will have a new origin located at the lower left corner of the part.

#n# CONS PAR VBLOCK ORG: MODEL ENT d1d2
d3d4d5d6d7;MODEL LOC END d8
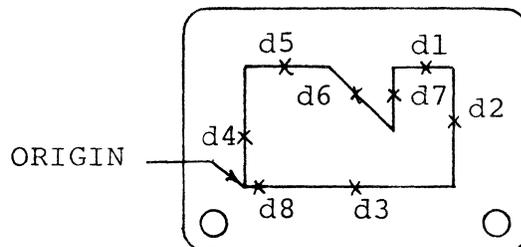
COMMAND	DEFINE CONSTRUCTION PLANE
DESCRIPTION	The Define Construction Plane command allows the user to define planes for later use by other commands. These cplanes may have any orientation and offset relative to the model coordinate system. These planes can then be used in defining views of the model as planes on which the views are to be projected. (See Define View command). These cplanes can also be used to determine the special orientation of entities inserted into the model, regardless of the orientation of the view in which the placement digitize was entered.
	When a new part is activated, the XY plane of model space serves as the default active construction plane for defining views, inserting entities, or digitizing. By use of the Select Construction Plane command, the user may select any other pre-defined cplane as the current active construction plane. In addition the system provides List, Delete, and Echo Construction Plane commands to allow listing and deleting defined cplanes, and activating display of cplane temporary graphics.
SYNTAX	DEF CPL name (modifiers):DRAW/MODEL LOC d1(d2d3)
MODIFIERS	
NAME1	Name1 = name of Cplane being defined, 20 characters maximum.
NORMAL	Defines a Cplane from a normal vector. First digitize locates the origin of the construction plane. First and second digitize defines the normal vector. Third digitize is projected on to the construction plane and defines the direction of the X-axis.
DIGITIZES	If only one digitize is given, it will be used for the origin. If no orientation modifier is given (FROM +name, DIMET +, or angle), then the XY model orientation is used. If three digitizes are given, the orientation is calculated using the first digitize as the origin, the second as a point on the positive X-axis, and the third as a point in the positive Y-plane (i.e., any point in the first or second quadrant).

The system makes available seven pre-defined
permanent standard cplanes, which can be
accessed and referenced by the user in the
same manner as any that might be defined,
except that they cannot be deleted. These
can be referenced either by their number,
1 to7, or interchangeably by their alternate
names as follows. (Note, however, that if
a user is in a two-dimensional part, there is
only one predefined cplane, 1=TOP=XY=MODEL.)

STANDARD CPLANE	ALTERNATE NAMES
1	TOP, XY, MODEL (default)
2	FRONT, XZ
3	RIGHT, YZ
4	BOTTOM, X-Y
5	LEFT, -YZ
6	REAR, -XZ
7	ISO

Note that Cplanes 1 to 6 are the six faces of
a cube. Cplane 1 (=TOP=XY=MODEL) is the
default cplane used by the system when none
other has been selected or otherwise referenced.
Cplane 7 or ISO is the isometric plane which
provides an equal projection of all three
axes. It can be envisioned as a cube balanced
on one corner.

ERROR MESSAGES

CPLANE NAME NOT SPECIFIED

No cplane name was input

ORIENTATION SPECIFICATION INCOMPLETE-
3 DIGITIZES REQUIRED

Command was terminated before the three expected
digitizes were input.

INVALID DATA- DIGITIZES DO NOT DEFINE
A UNIQUE

The diametric ratio must be between -2 and +2.

CPLANE NAME ALREADY IN USE

A cplane name was entered which is already
used by a previously defined cplane.

NO CPLANE EXISTS NAMED NNNN

The cplane referred to by the FROM or VIEW
modifiers does not exist.

#n#DEF CPL 8:MODEL LOC d1d2d3

COMMAND	DEFINE VIEW
DESCRIPTION	The Define View command creates a new view within an active drawing. A drawing may contain any number of views, each of which will occupy a two-dimensional rectangular area of the drawing. This region is referred to as the view clipping window. Graphics are not visible outside of this window. Thus entities that cross an edge of the window will be visible only up to the point of crossing. The two basic methods of creating views are explicit referenced to a defined construction plane, and folded auxiliary view specification.
SYNTAX	DEF VIE viewname CPL cplname (modifiers): DRAW LOC dl DRAW LOC d2 DRAW LOC d3
viewname	A required alphanumeric text string consisting of up to 20 characters. Special characters should not be used.
cplname	The name of the construction plane to be used in defining the view. The system for seven permanently defined cplanes TOP, (default), FRONT, RIGHT, BOTTOM, LEFT, REAR, and isometirc (ISO). Other construction planes may be user-defined using the Define Cplane command.
MODIFIERS	

SCALEn	Specifies a scale value for the view being created. The permissible formats of scale specifications are as follows:
	
	    1. Single numerical value SCALE 2.5
	
	    2. A ratio of two numbers SCALE 4 TO 2
	
	    3. A unit ratio of numbers SCALE 10 IN TO 1 KM
	
	The available unit designations are:
	
	    MM (millimeters) IN (inches)
	
	    CM (centimeters) FT (feet)
	
	     M (meters) MI (miles)
	
	    KM (kilometers)

XPn YPn ZPn	Coordinates used to specify a vantage point that controls the perspective appearance of the view. Each numeric value n defines the coordinate position of a vantage point. Coordinates are relative to the drawing coordinate system and are given in drawing units. Vantage point cooresponds to the eye position of a viewer. To establish a perspective view the value of ZP, a positive non-zero number, must be specified. If XP and YP are omitted, the origin of its current view is used.
PERSP	Simple way to establish a perspective vantage point. No coordinate values are required. It causes the X and Y coordinates of the vantage point to be set to the origin of the view, and the Z coordinate to be set to 64 cm.
DMIN,DMAX	Establishes the minimum and maximum extents of depth-clipping to be applied to the view (for depth clipping a positive value extends into the screen). The values are entered in model space units and are relative to the construction plane.
TILTA	Tilt angle 'A' in degrees counterclockwise; rotation is in the plane of the screen (default = $\emptyset$).
DIGITIZES	After all modifiers have been entered, the user enters a colon to begin digitizing. The system responds with the prompt DRAW loc and awaits the first digitize. This digitize specifies the point an the drawing where the views origin will be positioned. The system then proceeds to create the view and generate its display. (Note: For the first insertion of a view in a drawing, a lapse of time will occur before the image appears.) The view's image at this point is unclipped. The system will accept two additional digitizes which specifies opposite corners of the desired clipping window.
EXAMPLES	

EXAMPLE   #n#DEF VIE:VIEW d1d2d3: VIEW d4d5d6:VIEW d7d8d9

COMMAND	EXIT PART
DESCRIPTION	Exit Part command is used to exit from an active part and return to CADDS environment thereby accomplishing one of four functions:

1. Alloying the part to be filed before returning to the operating system.
2. Returning to operating system without filing.
3. Renaming and filing current part name.
4. Returning to CADDS environment without filing.

SYNTAX	EXIT PAR $\begin{array}{c} F \\ Q \end{array}$ newname OS (CR)

MODIFIERS

F Q	One of these modifiers must be used or the command will be ignored. F is used to file active part before EXITING; Q is used to quit the part without filing.
OS	Returns the user to the operating system. OS must be accompanied with an F or Q.
newname	Renames the active part and files it.
EXAMPLE	The following example would file the currently active part under the name BRACKET and return to the operating system level

        #n#EXIT PAR F BRACKET OS

ERROR MESSAGES	BAD PARTNAME

This message is output if the name exceeds 60 characters or contains illegal characters.

        F OR Q REQUIRED

This message is output if the F or Q modifiers are not given.

        PART EXISTS, TYPE OK TO OVERWRITE

This message is output if a part already exists with this name. OK allows the file to be up-dated with any changes made since the last filing.

COMMAND	FILE PART
DESCRIPTION	The File Part command files the current version of an active part under a new name, the current name (default), or files the TVFs. A part may be filed any time it is active. Upon completion of the filing, the user may continue modifying the active part until EXITing or returning to the operating system.
SYNTAX	FILE PARt modifiers (CR)
MODIFIERS	
newname	New name of filed part. Must be less than or equal to 60 characters.
NOTE	Only an active part can be filed.
ERROR MESSAGES	BAD PARTNAME
	Displayed when name contains more than 60 characters or an illegal character.
	XXX EXISTS, TYPE OK TO OVERWRITE
	Displayed if a part already exists with the new name. OK allows the file to be updated with current information.

DATABASE CONSTRUCTION COMMANDS

Database Construction commands are used to create and to edit the graphical entities that become the model in the part database. These graphical entities include points, lines, arcs, text, etc. Graphical entities may also be created and stored in the drawings of a part. These commands include:

```
CHANGE TEXT
CONSTRUCT GROUP
CONSTRUCT OFFSET
DELETE ENTITY
DIVIDE ENTITY
INSERT ANGULAR DIMENSION
INSERT ARC
INSERT B-SPLINE
INSERT CHAMFER
INSERT CIRCLE
INSERT CENTER LINE
INSERT DIAMETER DIMENSION
INSERT ELLIPSE
INSERT FILLET
INSERT FLAG
INSERT LABEL
INSERT LINEAR DIMENSION
INSERT LINE
INSERT POINT
INSERT TEXT
INSERT CROSSHATCH
MIRROR ENTITY
ROTATE ENTITY
STRETCH ENTITY
TRANSLATE ENTITY
TRIM ENTITY
```

COMMAND	CHANGE LAYER
DESCRIPTION	The Change Layer command changes the layer associated with selected entities. All selected entities will be moved from their original layer to the specified layer
SYNTAX	CHA LAYn:MODEL ENTd1d2...dn
MODIFIER	
n	n is an integer between 0 and 254, which specifies the layer onto which the selected entities will be moved.
RESTRICTION	The layer on which an entity is located must be active or echoed (i.e., the entity must be visible) before that entity can be changed to another layer.
EXAMPLE	Assume two entities are on the screen: a line on layer 5 and a circle on layer 10.

#n#CHA LAY20:MODEL ENTd1d2
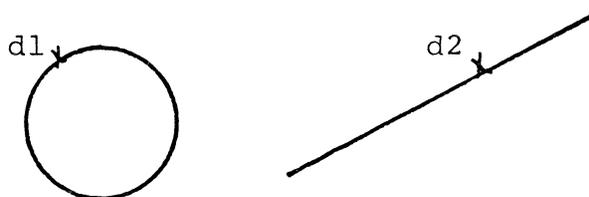
Both the line and the circle will be moved to layer 20.

COMMAND	CHANGE TEXT
DESCRIPTION	The Change Text command is used to change the text parameters of existing entities of the following types:

Text
Nodal Text
Linear Dimensions
Angular Dimensions
Radial Dimensions
Diameter Dimensions
Feature Control Symbols
Labels

SYNTAX	CHA TEX (modifiers): DRAW/MODEL ENT d1d2...dn
MODIFIERS	
HGTn	Changes the text height to the value n.
PENn	Changes PEN code to this value.
FONTn	Changes FONT code to this value.
LJT,CJT,RJT	Specifies left-, center- or right-justification.
MN	Specifies no mirroring.
MX,MY,MB	Specifies mirroring about the XZ plane, YZ plane or both planes.
THKn	Changes text aperture thickness to the value n. This modifier specifies what happens when the text is sent to a photoplotter or any other plotter which has the capability to plot thickness. No change is visible on the screen display.
WDTn	Changes text character width to the value n.
SLTn	Changes text character angle slant to the value n.
ANGn	Changes text base rotation to the value n.
RATIOn	Changes the text height and width uniformly by the given ratio. It must, therefore, be used exclusively of height and width.

STRNG'x'   Replaces the existing string with the string
                specified. The new string must be closed by
                single quotes (').

     NOTE   For text with In-Stream modifiers, use
                     the Edit Text command.

 EXAMPLES   #n#CHA TEXT STRNG'NEW':MODEL ENT dl

                            OLD
                           X dl
                          BEFORE

                            NEW

                           AFTER

COMMAND	CONSTRUCT GROUP

DESCRIPTION

The Construct Group command is used to create groups compiled of any number of entities. Its purpose is to provide a means by which entities can be quickly identified in GETDATA. All members in a group can be manipulated as a single entity by using the GETDATA modifier group. Entities contained within a gruop can become members of another group using the Group modifier.

It is not possible to add members to an existing group. If desired it must be accomplished by dissociating the group and restructuring it with new entities. Entities belonging to a group may not be deleted until the group is dissociated.

SYNTAX

CON GRO: DRAW/MODEL ENT d1d2....dn

RESTRICTIONS

A group may consist of either drawing or model entities but may not consist of both.

ERROR MESSAGES

GROUPED ENTITY, CONNOT DELETE

Displayed when the user tries to delete an entity belonging to a group.

EXAMPLES

#n#CONS GRO:MODEL ENT d1d2d3d4d5d6d7d8d9

COMMAND	CONSTRUCT OFFSET
DESCRIPTION	The Construct Offset command allows the user to construct an offset of a planar entity. The entity may be a point, line, arc, conic, B-Spline, string, or nodal line. In general, the offset of a point, line, arc, string, or nodal line will be the same respective entity. The offset of a conic or B-Spline is a B-Spline which approximates the true offset within the specified tolerance. A tapering capability exists which can be utilized to taper the offset along the specified curve. When this option is used, an arc, conic, or B-Spline will generate a B-Spline as the tapered offset. This command is legal in both model and drawing modes.
	In addition, the Construct Offset command also accepts all surfaces for offsetting.
SYNTAX	CONS OFF (modifiers) : MODEL ENT d1; MODEL LOC d2d3 (CR)
MODIFIERS	
D	The modifier D followed by a number specifies that the offset is to be that number of units displaced from the original contour. The default is .1. This modifier applies to surfaces also.
DEND	The modifier DEND followed by a number speciifes a tapered offset starting at a spacing D from the contour and gradually tapering to DEND at the end of the contour. The tapering of the offset is directly proportional to the length along the contour. Does not apply to surfaces.
STPT	The Starting Point modifier indicates that the user will digitize a point which will be projected onto the first curve of the contour. This projected point will serve as the starting point for offset curve. Does not apply to surfaces.
ENDPT	The same as STPT, except that it determines ending point for the offset.

ent

The prompt requests the user to select the entity to be offset.

loc

The prompt loc requests the user to choose the side of the curve for which the offset will be constructed. This prompt is also used when the start and/or end point modifiers are given to prompt the user to identify the start and/or end of the offset.

EXAMPLES

#n#CONS OFF D.25:MODEL ENT d1 MODEL LOC d2

x d2

x
d1
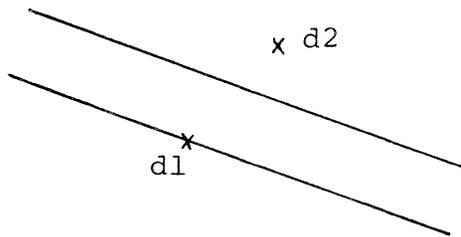
#n#CONS OFF D.25 DEND.5:MODEL ENT d1 MODEL LOC d2

d2
x

x
d1
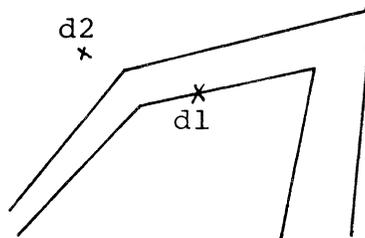
#n#CONS OFF D.25 DEND.5 STPT ENDPT:MODEL ENT d1
   MODEL LOC d2d3d4

d1 x

x----   --x
d3   d4

x
d2
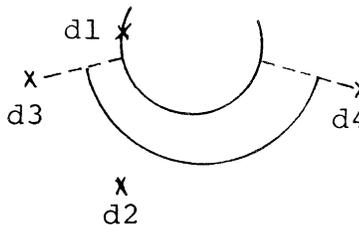
COMMAND   DELETE ENTITY

DESCRIPTION   The Delete Entity command is used to delete
                   graphical entities from the database. It
                   applies for both model and drawing entities.
                   Deleted model entities have their appearances
                   removed from all views in all drawings in
                   the part.

                   Delete differs from Erase in that deletion
                   means total entity removal from the part
                   whereas erasure specifies the removal of
                   an entity from a particular view of a par-
                   ticular drawing in the part.

SYNTAX   DEL ENT: MODEL ENT d1d2...dn (CR)

EXAMPLES   #n#DEL ENT:MODEL ENT d1d2d3

BEFORE

AFTER

COMMAND	DIVIDE ENTITY
DESCRIPTION	The Divide Entity command divides a given entity into separate smaller entities of the same type. The command can be used with lines, arcs, cpoles, conics or B-Splines. The divisions of the entity are designated either by explicit input of the number of divisions or by digitizing the division boundary points. At the termination of the command the original entity is deleted and is replaced by the newly formed entities.
SYNTAX	DIV ENT (modifiers):MODEL ENT d1; MODEL LOC d2...dn
MODIFIERS	
NDIVn	Number of divisions into which the entity has to be divided.
ent	User should digitize the entity to be divided.
loc	This prompt will appear if no modifier is input (or NDIV=0). The user has to digitize the division boundary points. (However, the start of the entity is assumed to be the start of the 1st entity division and the end of the entity to be the end of the last division and, hence, need not be digitized). Also, if the digitizers are not in order along the entity, they will be sorted before being considered for division boundary points.
RESTRICTIONS	1. Only one entity can be selected at a time.
	2. Digitizes lying beyond the ends of the entity are rejected.
	3. A maximum of 50 digitizes can be entered.
ERROR MESSAGES	MAXIMUM DIGITIZES REACHED
	If more than 50 digitizes are selected this message will be printed. The first 50 digitizes will be retained and the command will be completed.

MORE THAN ONE ENTITY SELECTED; RESELECT

If more than one entity is selected, this
error message will be printed and the user
will have to reselect the desired entity again.

EXAMPLES

#n#DIV ENT:MODEL ENT d1 MODEL LOC d2d3d4d5
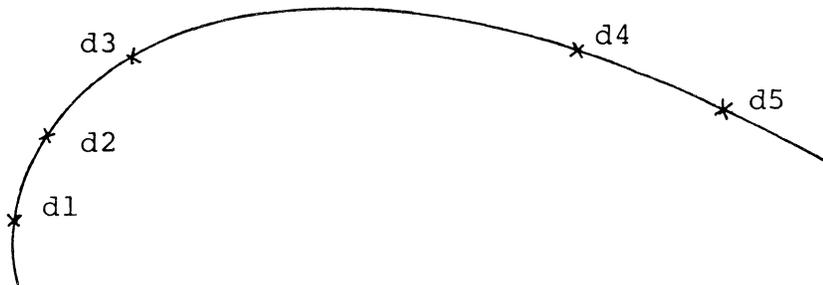
#n#DIV ENT NDIV8:MODEL ENT d1
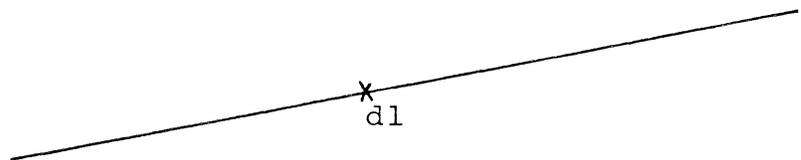
COMMAND	INSERT ANGULAR DIMENSION
DESCRIPTION	Angular dimensions measure the angle between two lines. The angle is specified by treating the two lines as vectors and identifying the terminal end of each vector. The angular dimension measures the angle between the terminal ends of the vectors. A third digitize specifies the palcement of text. The command automatically inserts extension lines (if needed), curved dimension lines, arrowheads, and dimension text.

Angular dimensions are associative by default. That is, the dimension is related to the lines being dimensioned and will automatically be re-created ("regenerated") when the lines are modified.

<div style="text-align:right"></div>

SYNTAX   INS ADI (modifiers): DRAW/MODEL ENT-END d1d2
DRAW LOC d3

Three modes of operation are available: standard mode, align, and rangle. Extension lines will be automatically drawn from the terminal ends of the lines defining the angle (whenever appropriate).

STANDARD MODE:
      INS ADI (modifiers): DRAW/MODEL ENT-END d1d2
      DRAW LOC d3
Where
      d1 and d2 select the line segments
      describing the angle and d3 locates the
      text origin.

ALIGN MODE:
      INS ADI TEXT LOCATION ALIGN (modifiers):
      DRAW/MODEL ENT-END d1d2 DRAW ENT d3 DRAW LOC d4
Where
      d1 and d2 select the line segments
      describing the angle to be dimensioned,
      d3 selects the existing angular dimension
      to be used for alignment, and d4 locates
      the text origin.

RANGLE X:
      INS ADI RANGLE X (modifiers): DRAW/MODEL
      ENT-END d1 DRAW/MODEL ENT-END d2 DRAW LOC d3
Where
      d1 selects the first line segment

describing the angle and d2 selects the point which, along with the angle of X degrees determines the second line (extension line) describing the angle, and d3 locates the text origin.

MODIFIERS

LEADER

Specifies leader parameters according to the following second-level modifiers:

    HEAD   The leader is drawn to the head of the text.
    END   The leader is drawn to the end of the text.
    CENTER   The leader is drawn toward the center of the text.

RANGLE

Specifies that hte second line in the angle to be dimensioned is a reference line at the given angle. The first digitize identifies the terminal end of a line (as usual). The second digitize fixes the reference line by specifying a point that it goes through. The third digitize specifies a test location (as usual).

SUPPRESS

Suppresses extension lines (first level) according to the second-level modifiers which follow:

    FIRST   The first extension line is suppressed. Extension lines are suppressed in the order of the digitizations.
    SECOND   The second extension line is suppressed.
    BOTH   Both extension lines are suppressed.
    NEITHER Neither extension line is suppressed. This is the default.

ERROR MESSAGES

The following error messages are self explanatory and may be displayed for informative purposes only:

    ARROWS FLIPPED TO INSURE VALID DIMENSION
    TEXT MOVED TO INSURE VALID DIMENSION
    MODE OVERRIDE NOT ALLOWED. REENTER DATA
    ALIGNMENT DIMENSION IS NOT COMPATIBLE
    ENTITIES TO BE DIMENSIONED MUST BE IN THE
        SAME VIEW
    ENTITIES TO BE DIMENSIONED MAY NOT BE
        PARALLEL, CO-PLANAR, OR ZERO LENGTH

EXAMPLES

#n#INS ADI:MODEL ENT-END d1d2 MODEL LOC d3
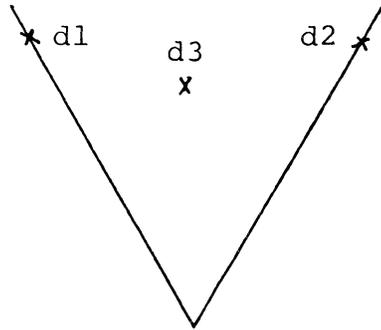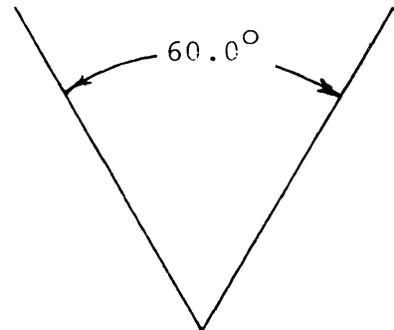
BEFORE   AFTER

#n#INS ADI RANGLE0:DRAW/MODEL ENT-END d1 DRAW/
    MODEL ENT-END d2 DRAW LOC d3

BEFORE

AFTER

COMMAND	INSERT ARC

DESCRIPTION

The versatile command Insert Arc can be used to insert an arc by:

1. Digitizing its begining, mid and end points.
2. Projecting an angle for its begining and end points.
3. Explicitly inputting a desired angle for its begining and end points.

In all an arc can be inserted 31 different ways using various modifier combinations.

SYNTAX

INS ARC (modifiers):MODEL LOC d1d2...dn

MODIFIERS

AGO/AGOn

Starting angle of arc; may be used with or without explicit value n (default 0).

AEND/AENDn

Ending angle of arc, used with or without explicit value n (default 0).

RAD/RADn

Radius of arc, used with or without explicit value n.

DIAM/DIAMn

Diameter of arc, used with or without explicit value n.

NOTE

Arc is always inserted counterclockwise in any one of the 31 ways that follow:

1.	INS ARC: LOC ddd	ddd=3 pts on circum of arc
2.	INS ARC AGOx: LOC ddd	ddd=3 pts on circum of arc AGO explicit, d3=AEND
3.	INS ARC AENDx: LOC ddd	ddd=3 pts on circum of arc AEND explicit, d1=AGO
4.	INS ARC AGOx AENDx: LOCddd	ddd=3 pts on circum of arc AGO & AEND explicit
5.	INS ARC AGO: LOC dddd	ddd=3 pts on circum of arc d4=AGO

```
25. INS ARC DIAM AENDx:LOC dd      dl & d2=DIAM
                                   dl=AGO, AEND explicit
26. INS ARC DIAM AGOx AENDx:        dl & d2=DIAM
    LOC dd                          AGO & AEND explicit
27. INS ARC DIAM AGO: LOC ddd       dl & d2=DIAM
                                   d2=AEND, d3=AGO
28. INS ARC DIAM AEND: LOC ddd      dl & d2=DIAM
                                   dl=AGO, d3=AEND
29. INS ARC DIAM AGO AEND:          dl & d2=DIAM
    LOC dddd                        d3=AGO, d4=AEND
30. INS ARC DIAM AGOx AEND:         dl & d2=DIAM
    LOC ddd                         d3=AEND, AGO explicit
31. INS ARC DIAM AGO AENDx:         dl & d2=DIAM
    LOC ddd                         d3=AGO, AEND explicit
```

NOTE

'x' after a modifier above indicates that the
user types in a value for AGO, AEND, RAD, or
DIAM after it.

EXAMPLES

The following are examples using some of the
various ways to input an arc.

#n#INS ARC:MODEL LOC dld2d3
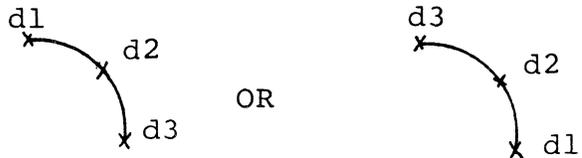
#n#INS ARC AGO45:MODEL LOC dld2d3
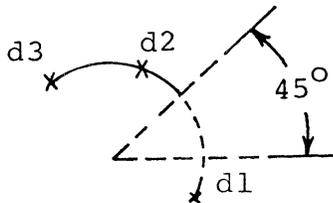
#N#INS ARC AEND:MODEL LOC dld2d3d4
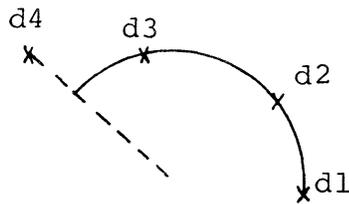
#n#INS ARC AGO AEND225: MODEL LOC d1d2d3d4
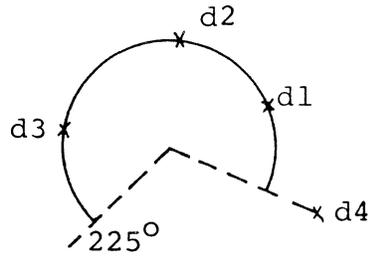
#n#INS ARC RAD2:MODEL LOC d1d2d3
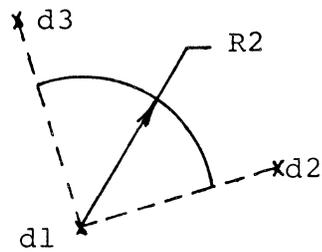
#n#INS ARC DIAM3 AGO270 AEND45:MOSEL LOC d1
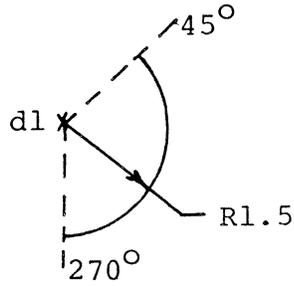
#n#INS ARC RAD: MODEL LOC d1d2d3
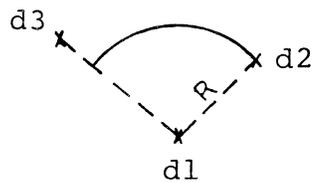
#n#INS ARC RAD AGO90:MODEL LOC d1d2
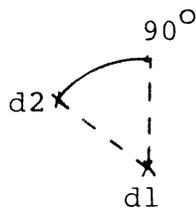
#n#INS ARCRAD AGO AEND:MODEL LOC d1d2d3d4

d3

d2

R

d1

d4
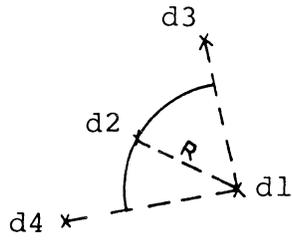
#n#INS ARC DIAM:MODEL LOC d1d2

d2

d1
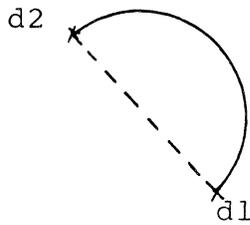
#n#INS ARC DIAM AGO90:MODEL LOC d1d2

90°

d2

d1
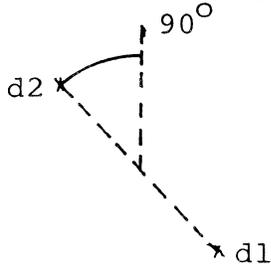
#n#INS ARC DIAM AEND:MODEL LOC d1d2d3

d2

d1

d3
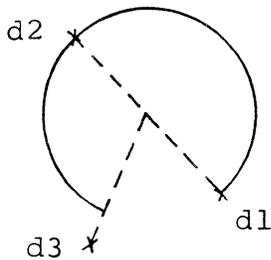
#n#INS ARC DIAM AGO AEND45:MODEL LOC d1d2d3

45°

d2

d3

d1

COMMAND	INSERT B-SPLINE
DESCRIPTION	The Insert B-Spline command is used to insert B-Splines by the digitizing of two or more points. (A B-Spline between two points is a straight line.)
	CADDS allows the insertion of second through seventh degree (Quadratic) or third degree (Cubic) B-Splines. The Insert Bspline command however, uses the default of a non-rational B-Spline definition, i.e., for each function in the inserted sequence H (t) = 1.
	The ORIGIN of a B-Spline is the closest segment and point to the digitize. Valid in Model and Drawing modes.
SYNTAX	INSERT BSPLINE modifiers:
MODIFIERS	
DEG n	Takes an integer value (2-7) which allows the specification of frame one to seven degrees of the inserted B-Spline.
DEG2 (Quadratic)	A quadratic B-Spline ensures up to the first degree of derivative continuity, i.e., continuous tangent vectors slope.
DEG3 (Cubic)	A cubic B-Spline ensures up to the second degree of derivative continuity, i.e., continuous tangent vectors slope and continuous rate of change in slope.
	Default is DEG 3 (CUBIC).
	In general all other B-Splines have the Degree minus 1 continuous derivatives.
ZTN	A B-Spline is a 3-dimensional spline curve specified by the input of a sequence of 3 dimensional points. The Z component of these input points can be automatically overridden by a user-defined constant depth. This option will produce a planar B-Spline.
	The modifier ZT takes a real value (i.e., ZT3.62) which specifies a constant depth in the active construction plane for all input points. That is, the Z component of all points issued after the command colon (:) will be set to the value of ZT.
	Default is 3 dimensional.
INTR POLY	The modifiers INTR (interpolation) and Poly (polygon) set the mode of the user interface for definition of a B-Spline curve.
	Default is INTR.

3-391

POLY	Specifies that the digitized points are interpreted as defining POLYGON points.
INTR	Specifies that the digitized points are interpreted as curve points to which the B-Spline will interpolate. There are three forms of command syntax with the modifier INTR depending upon the input of two other modifiers TANA and TANB.
NOTE	INTR is the default mode if modifier POLY is not input.
	Under the interpolation mode, more information than a sequence of inter-polant points is needed to uniquely define the interpolating B-Spline. This added information can be specified by user determined boundary conditions at the ends of the spline. Boundary conditions consist of the input of start and/or end first derivative vectors (tangents).
TANA	Input start tangent vector.
TANB	Input end tangent vector.
CAUTION	Vector magnitude (i.e., length of tangent vectors) together with direction influences the interpolation.
NOTES	A quadratic B-Spline (DEG2) needs one boundary condition for its unique specification.
	All other higher degree B-Splines can take two boundary conditions.
CIRA CIRB	These modifiers are special cases of TANA and TANB respectively. Unlike TANA and TANB, they do not take user input. CIRA generates a start tangent which gives the B-Spline the initial slope and curvature of a circular arc fitted to the first three interpolant points. Likewise, CIRB approximates circular interpolation at the end of the B-Spline.
	As with TANA and TANB, CIRA and CIRB are used only in interpolant mode. TANA and CIRA are mutually exclusive; likewise, TANB and CIRB.
APPEARANCE MODIFIERS	
FONT	Automatic fonting will be specified in the following modifiers.
NAME n	Specifies the selected font name.
n	n = the selected font name. (This variation is valid only after FONT.)
ACT	Specifies the active font is reselected.
DIAM n	The pipe font is selected. n is the diameter.

NONE	No font is selected.
FLST	Lists all font names.
WDT n	Specifies the width of the PC width font.
DLEN n	Specifies the length of the PC width font's dashes.
SLEN n	Specifies the length of the PC width font's spaces.
LJT RJT CJT	The PC width font is left-, right-, or center-justified with respect to its defining line.
CRVA CRVB	CRVA gives the B-Spline the initial slope of a selected entity. Likewise CRVB gives the B-Spline the ending slope of some user-designated entity. CRVA is a substitute for TANA and CIRA and the three are mutually exclusive, as are CRVB, TANB, and CIRB in like fashion. The slope is that of the entity and nearer the selection digitize.
ADJUST	Used when the interpolant points are badly distributed; i.e., the spacing between points differs substantially. When the ADJUST modifier is used, interpolant points are added to insure that the B-Spline is smooth.
TAG="NAME"	Gives a 'user=defined' tag name to the B-Spline specified. '=' is a system prompt, after which the user types in the name of the tag. The name may contain up to six alphanumerics. Each tag name may be used only once.
STG	Allows the user to substitute the vertices of a string or nodal line for explicit input of points. When using the POLY modifier these vertices define the B-Spline polygon. When using the INTR modifier (or default), the B-Spline passes through the vertices. In either case, the vertices are ordered in the definition of the string or nodal line.
EXAMPLES	Input of B-Spline POLYGON directly in an approximation mode.
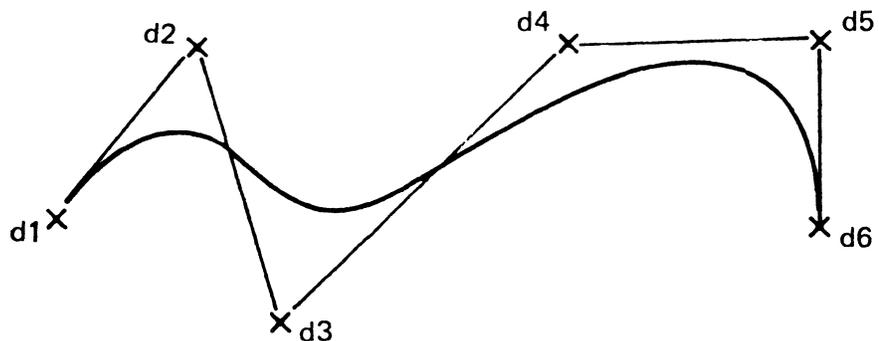
#n#INSERT BSPLINE POLY : loc d1 d2 d3 d4 d5 d6

EXAMPLE 2

Quadratic versus Cubic B-Spline

#n#INSERT BSPLINE DEG3: loc d1 d2 d3 d4 d5 d6

or

#n#INSERT BSPLINE: loc d1 d2 d3 d4 d5 d6
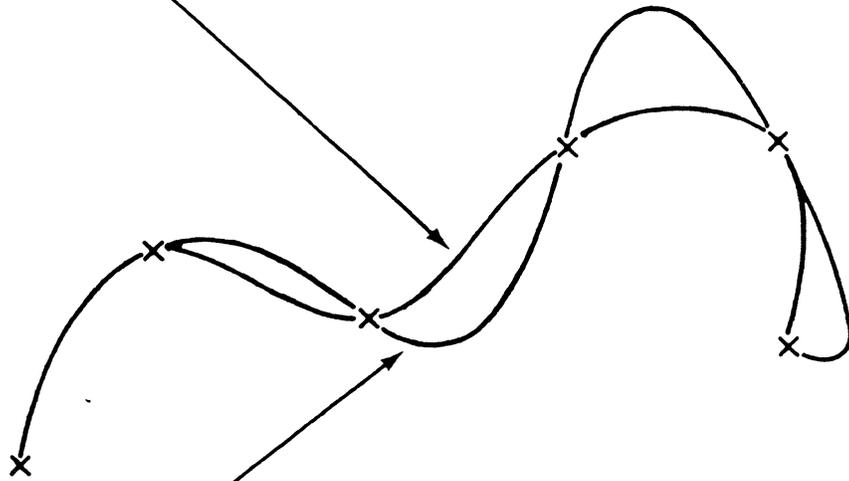
#n#INSERT BSPLINE DEG2: loc d1 d2 d3 d4 d5 d6

THE HIGHER THE DEGREE,
THE "SMOOTHER" THE CURVE

EXAMPLE 3

This example shows the difference between a quadratic (Degree 2) Bspline and a 5th degree Bspline.

TOP VIEW
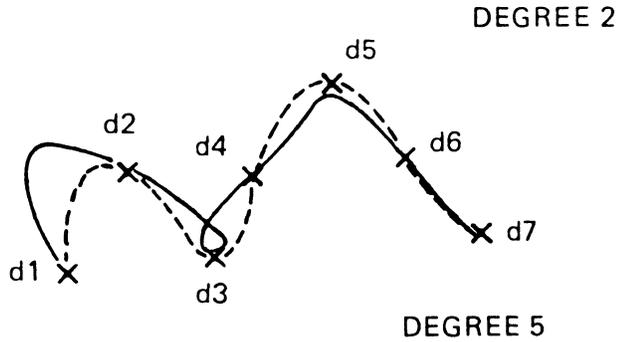
DEGREE 2

DEGREE 5

Degree 2 appears in solid lines.

Degree 5 appears as a dashed line.

SYNTAX

INSERT BSPLINE DEGREE 2: loc d1d2d3d4d5d6d7

INSERT BSPLINE DEGREE 5 FONT DASH: loc d1d2d3d4d5d6d7

EXAMPLE 4
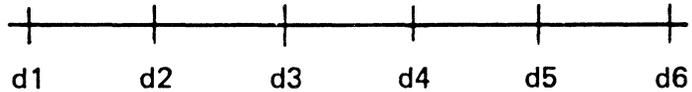
#n#INSERT BSPLINE: loc NT d1 d2 d3 d4 d5 d6

COMMAND	INSERT CHAMFER
DESCRIPTION	The Insert Chamfer command inserts a chamfer between two intersecting lines.
SYNTAX	INS CHA (modifiers): MODEL ENT d1d2 MODEL LOC d3
MODIFIERS	
ANG	Angle between line A and chamfer (default is 45.0).
LNGA	Distance from the intersection point of line A (first line chosen) to the end of the chamfer (default is 0.5).
LNGB	Distance from the intersection point of line B (second line chosen) to the end of the chamfer (default is 0.5).
TRIM	Trims lines to the end of the chamfer (default).
-TRIM	Suppresses trim
NOTE	If only the angle is entered, the user is prompted for a digitize along line A for the chamfer start point. If more than two out of LNGA, LNGB and ANG are entered, then the first two modifiers determine the chamfer.
ERROR MESSAGES	NO CHAMFER POSSIBLE
	If the resulting chamfer would have zero length, or be parallel to either line. No chamfer will be inserted.
EXAMPLES	#n#INS CHA:MODEL ENT d1d2
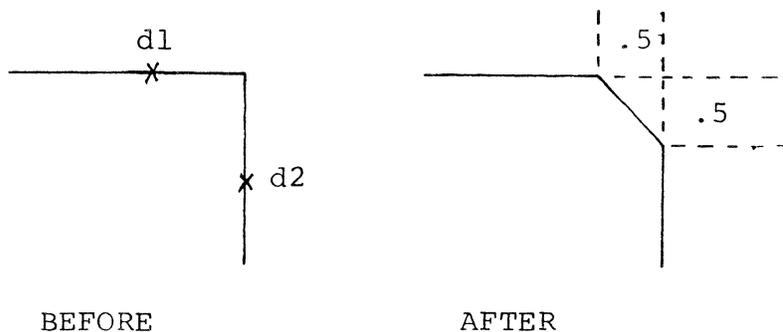
BEFORE   AFTER

#n#INS CHA LNGA.4 ANG30:MODEL ENT dld2

d1

d2

BEFORE

.4

30.0°
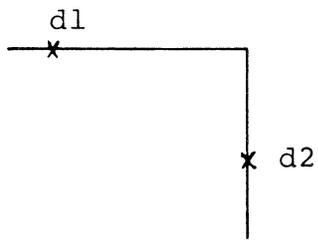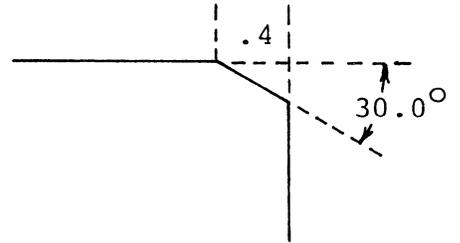
AFTER

#n#INS CHA -TRIM:MODEL ENT dld2

d1

d2
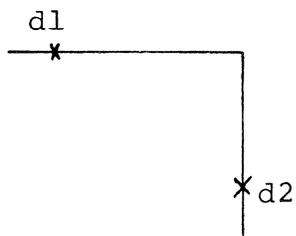
BEFORE
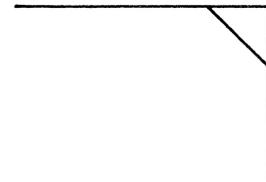
AFTER

COMMAND	INSERT CIRCLE

DESCRIPTION

The Insert Circle command provides several options for inserting a circle. The four basic methods of inserting a circle include digitizing:

1. The points on the circumference of the circle.

2. The center and a point on the circumference for the radius.

3. The center and input radius/diameter value explicitly.

4. Two points for a diameter from which the center and the radius will be computed. A circle endpoint is at zero degrees while its origin is its center.

SYNTAX

INS CIR (modifiers):MODEL LOC d1...dn (CR)

MODIFIERS

RAD

Radius of the circle (default is 0.0).

DIAM

Diameter of the circle (default is 0.0).

ARC

Creation of a circle from an existing Arc.

TANTO

Insertion of a circle, tangent to a line or a circle.

A circle may be inserted in one of the following ways:

1.	INS CIR: LOC ddd	3 pts on the circumference
2.	INS CIR RADx: LOC d	d=center, RAD explicit
3.	INS CIR DIAMx: LOC d	d=center, DIAM explicit
4.	INS CIR RAD: LOC dd	d1= center, d1-d2= radius
5.	INS CIR DIAM: LOC dd	d1-d2= diameter of circle
6.	INS CIR ARC: LOC d	d1= point on the arc

7. INS CIR TANTO: LOC dl ENT d2   dl=center d2=
                                      point on entity
      NOTE: 'RAD', or 'DIAM' have no function
            when used with 'TANTO' since the
            radius is described by dl and d2.
            If they are specified with 'TANTO',
            the modifier which appears last in
            the command line will be used.

#n#INS CIR:MODEL LOC dld2d3
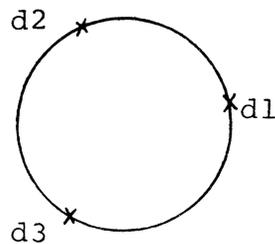
#n#INS CIR RAD1:MODEL LOC dl
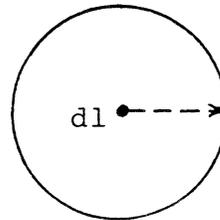
#n#INS CIR DIAM 1.5:MODEL LOC dl
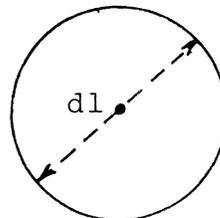
#n#INS CIR RAD: MODEL LOC d1d2

d2 •─ ─ ─ ─ ● d1
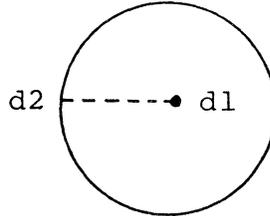
#n#INS CIR DIAM:MODEL LOC d1d2

d1

d2

#n#INS CIR ARC:MODEL ENT d1

d1

BEFORE   AFTER

COMMAND	INSERT CENTER LINE
DESCRIPTION	The Insert Centerline command generates a centerline between selected points or in selected circles. The cemterline consists of alternating long and short line segments, begining and ending with long segments. If the centerline is less than 0.3 inches, only the solid section of a centerline will be displayed.

Centerlines can be either line or circle entities. They are drawing entities by default unless the model modifier is used. |
SYNTAX	INS CLIN (modifiers): DRAW/MODEL ENT d1d2 DRAW/MODEL LOC d3...dn
MODIFIERS	
PNT	Point to point centerline (default).
CIR	Specifies two perpendicular centerlines through the center of the circle.
ANGn	Construct a linear centerline through a point at the specified angle n (in degrees).
CCIR	Construct a circular centerline defined by three points.
RADn	Construct a circular centerline defined by its center and radius n.
DIAMn	Construct a circular centerline defined by its center and diameter n.
MODEL	The centerline will be a MODEL entity. This can only be specified in the model mode.
EXAMPLES	#n#INS CLIN PNT:DRAW ORG d1d2 DEAW LOC(CR)
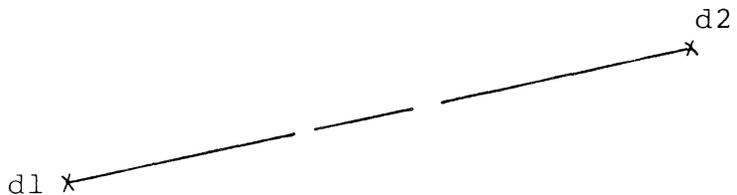
#n#INS CLIN PNT:DRAW ORG d1d2 DRAW LOC d3d4...dn (CR)

```
d1          d3              d4          d2
 X           X               X           X
```
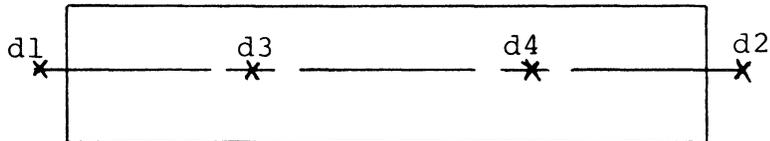
If more than two digitizes are given, the first
two indicate the ends, the remaining digitizes
locate the short segments.

#n#INS CLIN CIR:MODEL ENT d1

```
d1 X
```
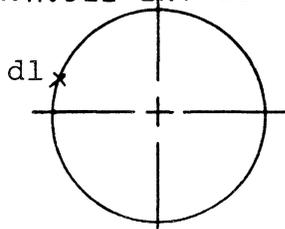
#n#INS CLIN RAD2.5:MODEL LOC d1

```
d1 +
```
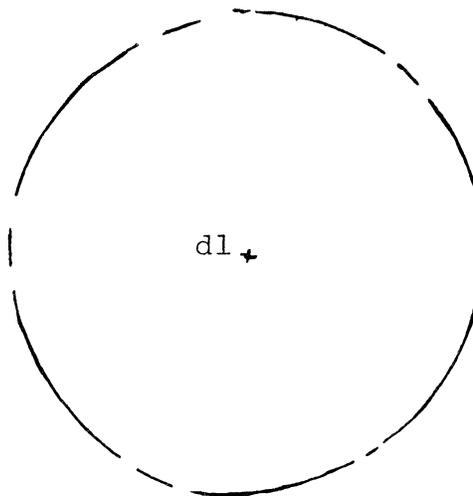
|	
COMMAND	INSERT DIAMETER DIMENSION
DESCRIPTION	The Insert Diameter Dimansion command is used to dimension the diameter of a circle and automatically inserts leaders, arrowheads, and dimension text. Diameter dimensions can be optionally associated with entities so that the dimension/ entity relationship is maintained as the entity undergoes transformation. This is done through a process called regeneration which requires the extraction of data from each related entity and the generation of an acceptable text origin. After this is done, the dimension is constructed in the same manner as when it was inserted. JIS, ISO, ANSI Standards are available via the Select Dimension command.
SYNTAX	INS DDI (modifiers): DRAW/MODEL ENT d1 DRAW/ MODEL LOC d2
RESTRICTIONS	1. Modifiers must be used in their hierarchical order or they will be ignored. 2. Either JIS, ISO, or ANSI standards can be used. They can not be used together.
ERROR MESSAGES	ARROWS FLIPPED TO ENSURE VALID DIMENSION ENTITY MUST BE PARALLEL TO A REAL VIEW TEXT MOVED TO ENSURE A VALID DIMENSION EXCESSIVE LEADER LENGTH DISALLOWS VALID DIM. HEAD/END CHANGED TO MIDDLE TO ENSURE VALID DIMENSION TEXT FLIPPED OUT TO ENSURE VALID DIMENSION
EXAMPLES	#n#INS DDI:MODEL ENT d1 DRAW LOC d2
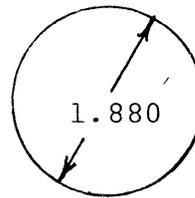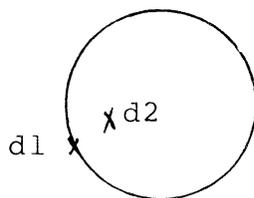
#n#INS DDI PRECISION PRIMARY 6: MODEL ENT d1
    DRAW LOC d2

d1 ✗   ✗ d2   1.879612
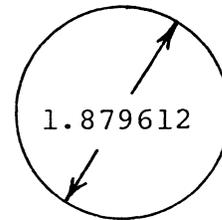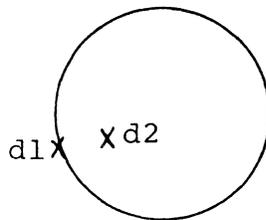

#n#INS DDI TEXT HGT .3:MODEL ENT d1DRAW LOC d2

d1 ✗   ✗ d2   1.880
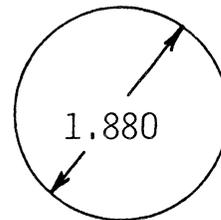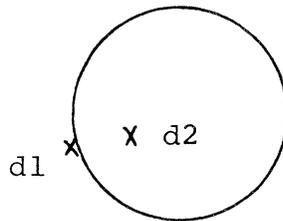
#n#INS DDI TEXT MAIN'3.56 MAX':MODEL ENT d1
    DRAW LOC d2

d1 ✗
   ✗ d2   3.56 MAX

COMMAND	INSERT ELLIPSE
DESCRIPTION	An ellipse is a second degree curve, oval in shape. It is inserted onto the active construction plane by one digitize for its center. The end point of a complete ellipse is zero degrees.
SYNTAX	INS ELL (modifiers): DRAW/MODEL LOC dl
MODIFIERS	
HMAJn	Half the length of the major axis. The major axis is the horizontal axis of an unrotated ellipse (default is 1.0 inch).
HMINn	Half the length of the minor axis. The minor axis is the vertical axis of an unrotated ellipse (default is 0.5 inches).
ANGAn	The true visual starting angle of the ellipse at the ellipse center (origin) with respect to the major axis (default is 0.0 degrees).
PANGAn	The parameterized starting angle of the ellipse at the ellipse center with respect to the major axis (default is 0.0 degrees).
ANGBn	The true visual ending angle of the ellipse at the ellipse center with respect to the major axis (default is 360 degrees).
PANGBn	The parameterized ending angle of the ellipse at the ellipse center with respect to the major axis (default is 360 degrees).
ROTn	The angle of rotation of the major axis from the horizontal (default is 0.0 degrees).
RESTRICTIONS	1. HMAJ and HMIN must be non-zero values with HMAJ greater than or equal to HMIN. 2. If modifiers referencing the same parameter are input, the last modifier typed is used.
NOTE	ANGA and PANGA refer to the same parameter. ANGB and PANGB also refer to the same parameter. 3. A zero curve length will default to a full ellipse.

#n#INS ELL:MODEL LOC X0Y0
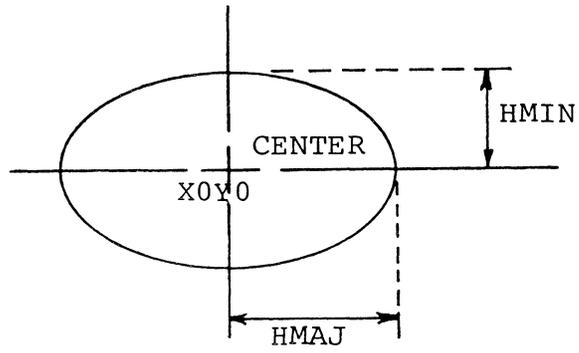
#n#INS ELL ANGA 23 ANGB -90 HMAJ 3 HMIN 2.2
    ROT 45:MODEL LOC X0Y1

COMMAND	INSERT FILLET
DESCRIPTION	A fillet is an arc with a specified radius inserted tangentially to two or three entities in the same plane. These entities can be any combination of point, line, arc, conic, B-spline, and string. The fillet is always inserted in the same plane as the entities.
SYNTAX	INS FIL (modifiers): DRAW/MODEL ENT d1d2(d3)
MODIFIERS	
RADn	Radius of the fillet (default is 0.5).
DIAMn	Diameter of the fillet (default is 1.0).
TRIM	Trim the two entities after the fillet is created (default mode).
-TRIM	Suppress trimming
TRIMA	Trim the first digitized entity only.
TRIMB	For two entity fillet trim the second digitized entity, for three entity fillet trim the third digitized entity.
DTRIM	Delay trimming. After the fillet is created and displayed the user is asked to digitize the end point of the entities that are to be trimmed.
CIR	Insert a circle instead of an arc.
LNGARC	Indicates the longer segment of the arc is wanted.
THRENT	Indicates three entity fillet. Three entities must be digitized.
ERROR MESSAGES	

FILLET UNDEFINED

Displayed if the fillet is not found between entities.

THE SECOND ENTITY MUST BE IN THE MIDDLE

Displayed if the second entity digitized,
using the modifier THRENT, is not located
in the middle of the three possible locations.

NO TRIMMING FOR FILLET ON SINGLE ENTITY

This message indicates that a fillet is being
fitted within the same entity (this is possible
for conic, B-spline and string). After the
fillet is created, the single entity will not
be trimmed.

EXAMPLES   #n#INS FIL:MODEL ENT d1d2

#n#INS FIL TRIMA:MODEL ENT d1d2

#n#INS FIL RAD.22 LNGARC:MODEL ENT d1d2

|	
COMMAND	INSERT FLAG
DESCRIPTION	The Insert Flag command provides the ability to create a stand-alone flag note complete with text.
SYNTAX	INS FLA "text" (modifiers): DRAW/MODEL LOC dl
MODIFIERS	
"text"	The text immediately following the noun FLAG is regarded as the main text, starting with currently selected font and case (See Select Text command), until altered by control characters. The entire text is assumed to be enclosed by a flag note indicator. The control characters to do this are automatically added.
HEIGHTn	Sets the character height as n data base units. The default character height is determined by the Select Text command.
MAIN't'	The entire main text (following the noun FLAG) is replaced by the text 't' following the MAIN modifier. The control characters to enclose the text in a flag note are automatically added. If other modifiers are to follow, this string must be delimited by quotes.
APPENDED't'	The text following the modifier APPENDED is appended to the main text string or its substitute. The text here may have its own control characters. The control characters required for creating a flag note are not automatically added.
PREFIXED't'	Specifies the text following the modifier PREFIXED is prefixed to the main text. The text here may have its own control characters. The control characters required for creating a flag note are not automatically added.
ANGLEn	Sets the angle of the flag note to n degrees. Default is zero degrees.
NOTE	The point digitized will be the origin of the flag note indicator, i.e., the lower-left corner of the flag.

#n#INS FLAG "AZ09":DRAW LOC dl

AZ09 dl
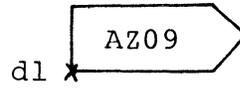
#n#INS FLAG "DEG" ANGLE 45:DRAW LOC dl

DEG dl
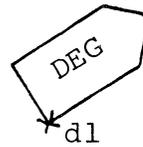
#n#INS FLAG "BIG" HEIGHT 1.5:DRAW LOC dl

BIG dl
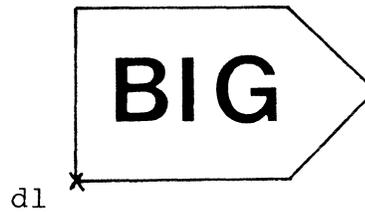
```
6.  INS ARC AEND: LOC dddd        ddd=3 pts on
                                      circum of arc
                                      d4=AEND
7.  INS ARC AGO AEND:LOC ddddd    ddd=3 pts on
                                      circum of arc
                                      d4=AGO,d5=AEND
8.  INS ARC AGOx AEND: LOC dddd   ddd=3pts on
                                      circum of arc
                                      AGO explicit,
                                      d4= AEND
9.  INS ARC AGO AENDx: LOC dddd   ddd=3 pts on
                                      circum of arc
                                      d4=AGO, AEND
                                      explicit
* * * * * * * * * * * * * * * * * * * * * * * * * * * *
10. INS ARC RADx/DIAMx: LOC ddd   d1=center, d2=AGO
                                      d3=AEND, RAD or
                                      DIAM explicit
11. INS ARC RADx/DIAMx AGOx:      d1=center, d2=AEND
    LOC dd                           RAD/DIAM & AGO
                                      explicit
12. INS ARC RADx/DIAMx AENDx:     d1=center, d2=AGO
    LOC dd                           RAD/DIAM & AEND
                                      explicit
13. INS ARC RADx/DIAMx AGOx       d1=center RAD/DIAM,
    AENDx: LOC d                     AGO,AEND explicit
* * * * * * * * * * * * * * * * * * * * * * * * * * * *
14. INS ARC RAD: LOC ddd          d1=center, d1-d2=RAD
                                      d2=AGO, d3=AEND
15. INS ARC RAD AGOx: LOC dd      d1=center, d1-d2=RAD
                                      d2=AEND, AGO explicit
16. INS ARC RAD AENDx:LOC dd      d1=center, d1-d2=RAD
                                      d2=AGO, AEND explicit
17. INS ARC RAD AGOx AENDx:       d1=center, d1-d2=RAD
    LOC dd                           AGO & AEND explicit
18. INS ARC RAD AGO: LOC ddd      d1=center, d1-d2=RAD
                                      d2=AEND, d3=AGO
19. INS ARC RAD AEND: LOC ddd     d1=center, d1-d2=RAD
                                      d2=AGO, d3=AEND
20. INS ARC RAD AGO AEND:         d1=center, d1-d2=RAD
    LOC dddd                         d3=AGO, d4=AEND
21. INS ARC RAD AGOx AEND:        d1=center, d1-d2=RAD
    LOC ddd                          d3=AEND, AGO explicit
22. INS ARC RAD AGO AENDx:        d1=center, d1-d2=RAD
    LOC ddd                          d3=AGO, AEND explicit
* * * * * * * * * * * * * * * * * * * * * * * * * * * *
23. INS ARC DIAM: LOC dd          d1 & d2= DIAM
                                      d1=AGO, d2=AEND
24. INS ARC DIAM AGOx: LOC dd     d1 & d2= DIAM
                                      d2=AEND, AGO explicit
```

COMMAND	INSERT LABEL
DESCRIPTION	The Insert Label command will insert text with leaders (lines) pointing to the labeled entity. The text must be proceeded and followed by a delimiter; usually quotes ("") though any special character may be used.
SYNTAX	INS LAB "text" (modifiers): DRAW/MODEL ENT dl DRAW/MODEL LOC d2...dn

MODIFIERS

LEADER	Specifies LEADER parameters according to its second-level modifiers.
ANGLEn	Sets the angle of the first segment of a leader with respect to the positive X axis. The command attempts to draw the first segment according to the following rules:

    1. Use LEADER ANGLE to determine the orientation of the first leader segment. If the specified angle causes the leader to miss the entity, try Step 2.

    2. Determine the closest point on the entity from the first digitization, and draw the leader to this point.

If the LEADER ANGLE modifier is not ised, the first segment of the leader will be drawn using step 2.

RADIAL	Causes the first leader segment to point toward the center if the selected entity is an arc or a circle.
HEAD	If, for the first leader to be specified, only two digitizations are entered, the leader will be drawn to the head of the text. This is the default condition.
END	If, for the first leader to be specified, only two digitizations are entered, the leader will be drawn to the end of the text.

ANGLEn	Text angle, where n is the angle in degrees.
HEIGHT't'	Label text height where n is the height. Default is global text height.
MAIN't'	Specifies that the main text is to be replaced by the given text 't'.
APPENDED't'	Specifies that the given text 't' will be appended to the main text. The maximum number of characters is 512.
PREFIXED't'	Specifies that the main text will be prefixed by the given text 't'. The maximum number of characters is 512.
EXAMPLES	#n#INS LAB "TEXT": MODEL ENT d1 MODEL LOC d2d3
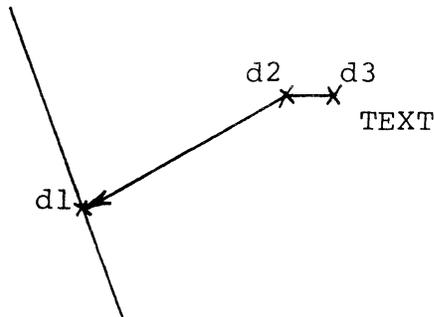
#n#INS LAB "TEXT":MODEL ENT d1 MODEL LOC d2
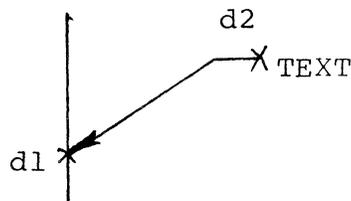
#n#INS LAB "TEXT" LEADER END:MODEL ENT dl DRAW
    LOC d2

```
                                    d2
                          TEXT  X──────►
                        │     ╱
                        │   ╱
                        │ ╱
                   dl  X◄
                        │
```
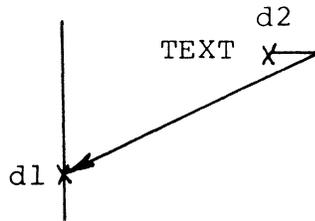
#n#INS LAB "(B5)FLAG(X)" LEADER END RADIAL:
    MODEL ENT dl MODEL LOC d2

```
              d2
     ┌──────┐ 
     │ FLAG │X─────
     └──────┘     ╲
                    ►─────╲
                      ╱        ╲
                    ╱   ○   X dl
                      ╲    ╱
```
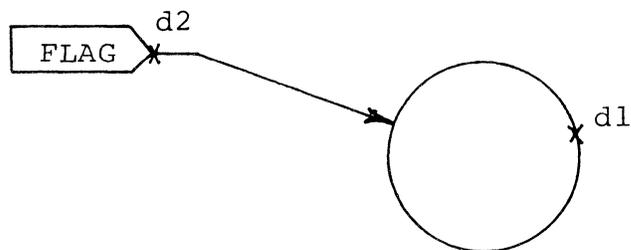
NOTE: 1. The (B5)...(X) control characters
         indicate a flag note.

      2. Because only two digitizations were
         used to define the label, the LEADER
         END modifier is necessary to force
         the leader to be attached to the end
         of the label.

      3. The leader is drawn radial to the
         circle because the LEADER RADIAL
         modifier was used.

      4. The last leader segment has a default
         length of 0.3.

COMMAND | INSERT LINEAR DIMENSION

DESCRIPTION

Linear dimensioning may be inserted between entities or points or intersections of entities. Two digitizations are required for the entities delimiting the distance being dimensioned and a digitization for the placement of the dimensioning text. JIS and ISO Dimensioning Standards are available via the Select Dimension command, in addition to ANSI Standards. The delimiting entities can be digitized in any order. The command automatically inserts extension lines, dimension lines and arrowheads, and dimensioning text. The three basic types of dimensioning are illustrated below:
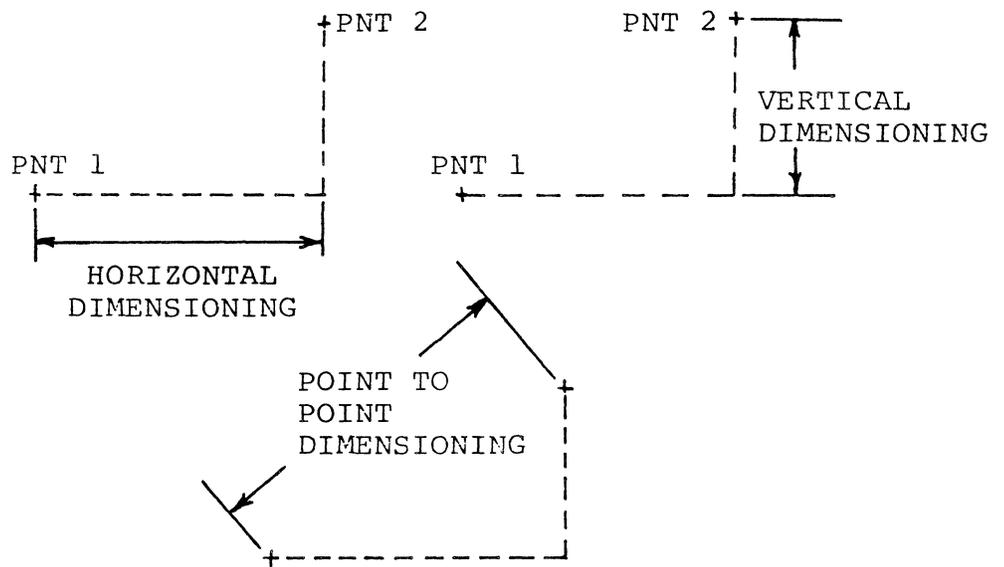
SYNTAX

INS LDI (modifiers): DRAW/MODEL ENT-END d1d2
DRAW LOC d3

MODIFIERS

CHAIN

Specifies that a chain of linear dimensions are created. It permits the user to select a number of points to be dimensioned and the location of the dimension lines. The dimension lines are co-linear and the text is automatically centered and aligned between the extension lines. The points to be dimensioned must be identified in the proper order.

HOR	Specifies that the horizontal distance between two points be measured (default).
PPOINT	Specifies that the dimension measures the straight line distance between two points. Extension lines are perpendicular to the dimension lines on the side of the dimension text.
SUPPRESS	This modifier suppresses extension lines according to the second-level modifiers which follow. Extension lines are suppressed in the order of the digitizations.
FIRST	The first extension line (first digitization) is suppressed.
SECOND	The second extension line (second digitization) is suppressed.
BOTH	Both extension lines are suppressed.
NEI	Neither extension line is suppressed. This is the default.
VER	The VERTICAL dimension measures the vertical distance between the arrowheads and places the extension lines horizontally.
ERROR MESSAGES	The following are status and error messages; the conditions are self explanatory.

      TEXT AND ARROWS FLIPPED OUT TO ENSURE
           VALID DIMENSION
      ARROWS FLIPPED TO ENSURE VALID DIMENSION
      HEAD/END CHANGED TO ENSURE VALID DIMENSION
      DIMENSION LINE MOVED TO ENSURE VALID
           DIMENSION
      DIMENSION LINE MOVED & TEXT AND ARROWS
           FLIPPED OUT TO ENSURE VALID DIMENSION
      DIMENSION LINE MOVED & ARROWS FLIPPED
           OUT TO ENSURE VALID DIMENSION
      DIMENSION LINE MOVED & HEAD/END CHANGED TO
           ENSURE VALID DIMENSION
      ARROWS FLIPPED OUT & HEAD/END CHANGED TO
           ENSURE VALID DIMENSION
      DIMENSION LINE MOVED, ARROWS FLIPPED OUT,
           & HEAD/END CHANGED TO ENSURE VALID
           DIMENSION

ENTITIES BEING DIMENSIONED MUST BE CO-
    LINEAR
MODE OVERRIDE NOT ALLOWED. REENTER DATA
XYZ PROPERTY DOES NOT EXIST REENTER DATA
ALIGNMENT DIMENSION IS NOT COMPATIBLE
ENTITIES TO BE DIMENSIONED MUST BE IN
    THE SAME VIEW
COINCIDENT POINTS NOT ALLOWED
AMBIGUOUS DIMENSION DEFINITION SPACE

EXAMPLES

#n#INS LDI:DRAW/MODEL ENT-END d1d2 DRAW LOC d3
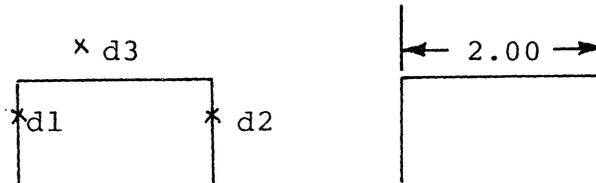
#n#INS LDI VER:DRAW/MODEL ENT-END d1d2 DRAW LOC d3

#n#INS LDI PPOI:DRAW/MODEL ENT-END d1d2 DRAW LOC d3
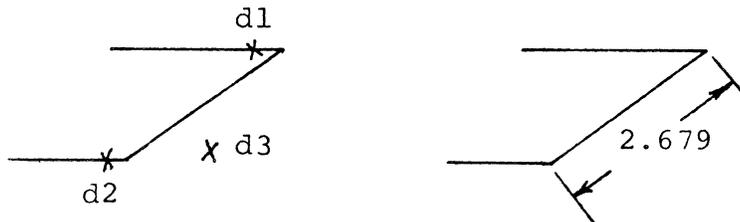
COMMAND	INSERT LINE
DESCRIPTION	The Insert Line command inserts a line between two specified points, each of which may be any X,Y,Z coordinate. The origin of the line is its midpoint.
SYNTAX	INS LIN (modifiers): DRAW/MODEL LOC d1d2...dn
MODIFIERS	
ANGn	Inserts a line at angle n with respect to a desired entity.
HOR	Inserts a horizontal line.
LNGn	Allows user to explicitly enter a desired line length (n).
PARAn	Constructs a line parallel to a given entity at a distance of n.
PERP	Constructs a line perpendicular to a specified entity.
TANTO	Inserts a line tangent to a specified entity.
VER	Inserts a vertical line.
ERROR MESSAGES	UNDEFINED

The message is most likely to occur when inserting a line tangent to a curve, using the modifier TANTO, and one of the following conditions has occured:

1. The curves have been specified at positions too far from the anticipated endpoints.

2. The line endpoints are inside the circle.

3. An attempt was made to insert a tangent line to a zero radius circle.

EXAMPLES	#n#INS LIN:MODEL LOC d1d2;MODEL LOC d3d4d5d6
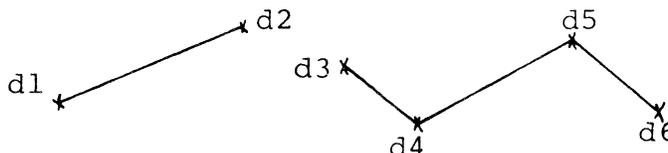
#n#INS LIN ANG45:MODEL ENT dl MODEL LOC d2d3

#n#INS LIN HOR:MODEL LOC dld2
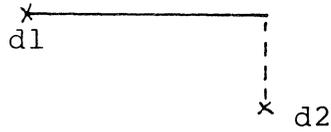

#n#INS LIN HOR LNG 5:MODEL LOC dld2
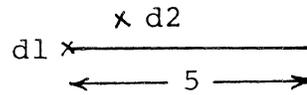
#n#INS LIN PAR 3:MODEL ENT dl MODEL LOC d2d3d4
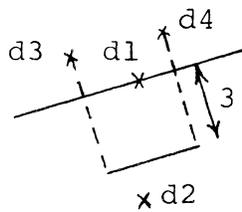
#n#INS LIN PERP LNG 5:MODEL ENT d1 MODEL LOC d2d3
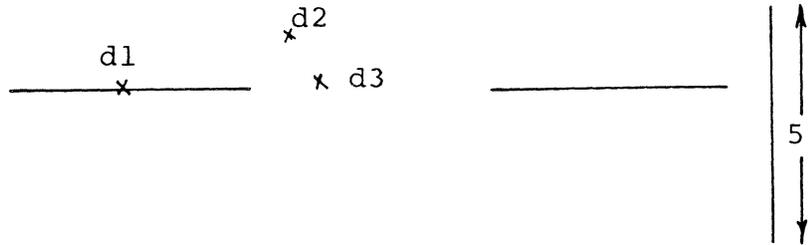
#n#INS LIN VER:MODEL LOC d1d2
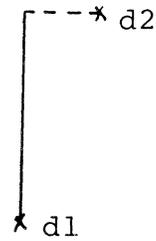
#n#INS LIN TANTO:MODEL ENT d1d2;MODEL ENT d3d4

COMMAND	INSERT POINT
DESCRIPTION	Insert Point is used to create points, either by specifying their coordinates or by specifying their position on an entity.
SYNTAX	INS POI (modifiers): DRAW/MODEL LOC d1d2...dn
MODIFIERS	
ON	Inserts a point on a selected entity (line, circle, arc, fillet, conic, spline, B-spline or tabulated cylinder by dropping a perpendicular in three spaces from a reference point to the entity. The point will be placed on the entity or on a proper extension of it. In drawing mode either a model or drawing entity may be selected for use with the ON modifier. If an entity from the model is selected, it is treated as if projected onto the drawing plane. The perpendicular is dropped in the drawing plane and a drawing space point is created.

Planar Entities

When inserting a point on a planar curve if the reference point is in the definition plane of the entity, then the inserted point is determined by dropping a perpendicular in the plane from the reference point to the entity. In other words if the line from the reference point to the inserted point were constructed, it would be perpendicular to a line tangent to the curve at the inserted point.

Three Space Entities

When inserting a point on a curve, if the reference point is not in the plane of definition of the entity, then the inserted point is determined by dropping a perpendicular in three space from the reference point to the entity. In other words, if the line from the reference point to the inserted point were constructed, it would be perpendicular to a line tangent to the entity at the inserted point.

|	
PROJ	Insert a point on a surface by dropping a digitized point onto the surface along a specified vector.

ent | Select entity for dropping perpendicular or for projection onto surface.

loc | Specify a coordinate location.

end | Specify the endpoint of an entity (used with PROJ modifier to specify projection vector).

EXAMPLES

#n#INS POI:MODEL LOC d1d2d3

d2 +

d1 +

d3 +

#n#INS POI ON:MODEL ENT d1 MODEL LOC d2d3d4d5
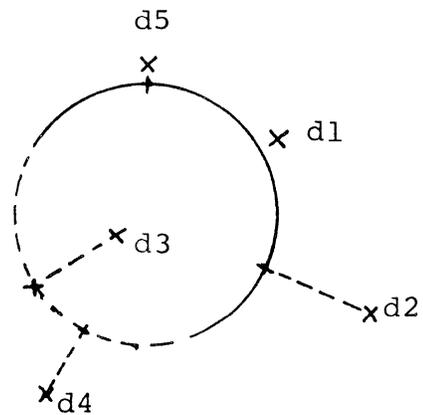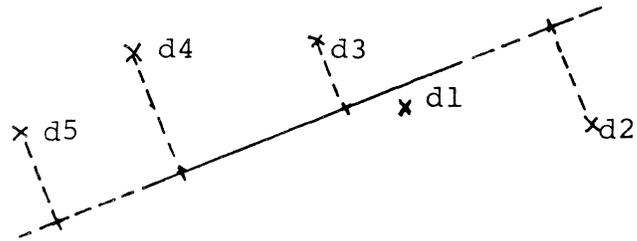
|	
COMMAND	INSERT RADIAL DIMENSION
DESCRIPTION	Radial dimensions may be inserted to dimension the radii of circles or arcs. The command requires one entity identification and an additional digitization for the placement of the dimension text. The command automatically inserts leaders, arrowheads, and dimension text.
	Dual dimensioning may be selected by the Select Dimension command. Dual dimensioning permits dimensions in two different unit systems (i.e., English and Metric) to be presented together as a single dimension.
	Radial dimensions are associative by default. This means that the dimension will be automatically regenerated (re-created) if the related arc is modified.
	The user may select either JIS, ISO, or ANSI dimensioning standards.
SYNTAX	INS RDI (modifiers):DRAW/MODEL ENT d1 DRAW/ MODEL LOC d2
EXAMPLES	#n#INS RDI:MODEL ENT d1 DRAW LOC d2

$\times$d2

$\times$d1

.50R
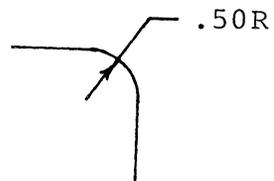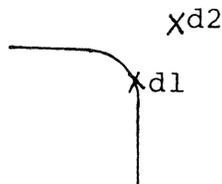
#n#INS RDI TEXT MAIN'2.765R':MODEL ENT d1 DRAW LOC d2

$\times$d2

$\times$d1

2.765 R
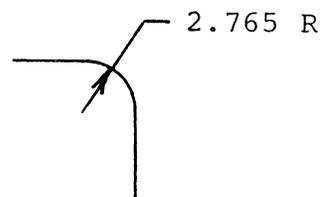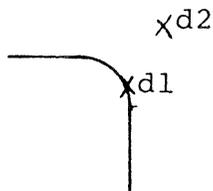
#n#INS RDIPRECISION PRIMARY 6:MODEL ENT dl
    DRAW LOC d2

X d2

Xdl

.387529 R

#n#INS RDI TEXT HGT .285:MODEL ENT dl DRAW LOC d2

X d2

X dl

.500R

#n#INS RDI LEADER END:MODEL ENT dl DRAW LOC d2

X d2

X dl

.500R

COMMAND	INSERT TEXT
DESCRIPTION	Text strings must be specified immediately following the verb-noun combination (INS TEX). Generally, each string is delimited on both sides by any character except A-Z, 0-9, colon, CNTRL-E or Rubout. All other characters (including space and carriage return) are permissible delimiters. If more than one string is specified, a double delimiter is used at the start and end of the series.
	If only one simple string is being specified, delimiters need not be given. Simple strings are those which begin with A-Z or 0-9 and contain no imbedded blanks, colons or quotes. (Any of these three will terminate the string.
	The order of text strings in the command corresponds to the order of insertion.
NOTE	There are several special cases for command syntax to be used with the Insert Text command to cover all the following contingencies:
	* Insertion of a standard character string
	* Insertion of a character string onto a series of text nodes
	* Auto insertion of a character string onto a series of text nodes (by specifying sequence number)
	* Auto insertion of a character string onto a series of text nodes contained in a subfigure (by specifying origin of subfigure)
	* Text character string insertion on nodal lines
SYNTAX	INS TEX text (modifiers):MODEL LOC d1d2...dn
NOTE	Insertion of a standard text string.
	The digitizations d(d...dn) specify the placements of the corresponding text strings in the series Text 1 (Text 2...Text n). If only one string is specified, it is replicated at each digitzed position.

INSERT TEX "PTI ISGREAT: MOD LOC

When it is desired to associate a text string to an auxiliary position (one or more), the digitization sequence

  ,dd1(d2...dn),

can be used in place of the ith digitization in the basic format. The text position is specified by d and the auxiliary position(s) are specified by d1(d2...dn). The sequence must begin and end with a comma.

Up to four auxiliary positions may be given.

MODIFIERS

LAYn	Selects layer for new entity where n is a number between 0 and 254. Default is current construction layer.
HGTn	Specifies text height.
WDTn	Specifies text character width.
THKn	Specifies text aperture thickness.
ANGn	Specifies text base rotation angle.
SLTn	Specifies text character slant angle.
FONTn	Specifies font code.
CJT	Specifies center-justified text (default).
RJT	Specifies right-justified text.
LJT	Specifies left-justified text.
MN	Specifies no mirroring (default).
MX	Specifies mirroring in X-axis.
MY	Specifies mirroring in Y-axis.
MB	Specifies mirroring in both X-axis and Y-axis.
LNSPn	Specifies interline spacing expressed as a percent of the text height value. Default ratio is 1.5.
DANGn	Specifies angle of text parallel with a designated line segment.

PRL	Specifies text parallel to a selected entity.
PRP	Specifies txt perpendicular to a selected entity.
OFFn.n	Specifies a perpendicular offset distance to an entity.
DIG	Specifies insertion of text at a digitized location.
FOR	Specifies a forward-reading text.
BACK	Specifies a backward-reading text.
FITL	Specifies fitting text within two digitized locations.
CURV	Specifies curved text.
INCR	Specifies the same type of name to several symbols.
IN-STREAM MODIFIERS	These modifiers are used within a text string to control subscripting, superscripting, font, slant, (italics), angle and case. These modifiers override basic modifiers specified by the Select Text command. While basic modifiers are in effect for the entire command, in-stream modifiers are in effect only for the remainder of the string in which they are specified.
	In-stream modifiers must be enclosed in delimiters. These delimiters must be different from those used to delimit the text string. The in-stream modifier delimiters are selectable via the Select Text command. The user may select different characters as the left and right delimiter or the same character may be used for both.
	A superscript follows. Only one level of superscripting is permitted, e.g. A B C is not valid.
	A subscript follows. If a subscript immediately follows a superscript, or vice versa, the superscript is positioned above the subscript.
E	End of subscript or superscript.

Fn	Font, where n is the font number.
	Fonts available are:

    1. CV Standard
    2. Leroy
    3. Undefined
    4. Microfont
    5. Lightline Gothic
    6. News Gothic
    7. Century Schoolbook
    8. Clearface
    9. Feature Control Symbols
  10. Calcomp Symbols
  11. Calcomp Text
  12. Experimental Symbols
  13. Reserved for customer font.
  14. Greek
  15. Helvetica
  16. ISO (Standard European)
  17. DIN17 (Standard European)
  18. MIL-STD
  19. Futura
  20. S**3
  21. Symbol Text

S	Slant (italics) begins
T	Slant ends.
An	Angle of text, where n is the angle desired.
U	Upper case begins.
L	Lower case begins.
EXAMPLES	#n#INS TEX ""TEXT"NOTE"R.25"":MODEL LOCd1d2d3d4

$d1_X$TEXT

$d2_X$NOTE

$d3_X$R.25

$d4_X$TEXT

|	
COMMAND	INSERT CROSSHATCHING
DESCRIPTION	The Insert Crosshatching command creates a crosshatching between the specified boundaries of the region to be crosshatched. Crosshatching lines are not independent of its boundaries; once generated, the lines move or change as the boundaries do.
SYNTAX	INS XHA (modifiers):DRAW/MODEL ENT d1d2...dn
MODIFIERS	
ANGLEn	Specifies the angle of crosshatching (in degrees). Any value, including a negative one, is permissible. Default is 45.0 degrees.
SPACINGn	Specifies the normal distance between the crosshatching lines. It must be positive. Default is 0.6cm = 0.236 in.
RESTRICTIONS	The maximum number of boundaries per crosshatch is 100. The number of entities in all boundaries must be 350 or less. Each crosshatching line can intersect the boundaries at no more than 100 positions. Boundary selection can be made either by separate digitizations or by use of the data modifier CHN.
EXAMPLES	#n#INS XHA:MODEL ENT d1d2d3d4
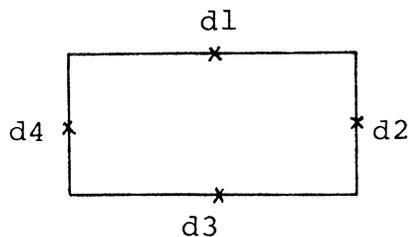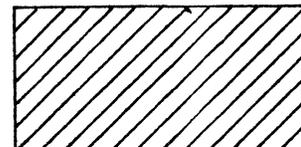
BEFORE   AFTER

#n#INS XHA:MODEL ENT CHN d1;MODEL ENT d2d3d4d5

d1

| O d2 | d3 O |
| O d4 | d5 O |
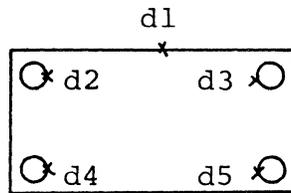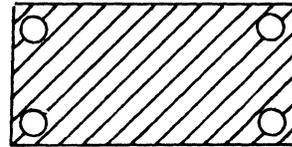
BEFORE   AFTER

#n#INS XHA ANG135 SPACING .3:MODEL ENT CHN d1;
      MODEL ENT d2d3d4d5d6

d1

| O d2 | d3 O |
| O d4 |
| O d5 | d6 O |

BEFORE   AFTER

SEE MODE DRA
  "   "   MOD.

COMMAND	MIRROR ENTITY
DESCRIPTION	The Mirror Entity command is used to mirror or mirror/copy entities by a reflective transformation across a plane in three spaces. The command is valid in both model and drawing modes.

        1. Selection of the entity or entities
           to be mirrored.

        2. Definition of the mirror plane.

A mirror plane can be defined as follows:

      Selecting one line as a mirror axis.

          The mirror plane is then defined to
          be perpendicular (normal) to the view
          of construction, through this line.

|	
SYNTAX	MIR ENT (modifiers):DRAW/MODEL ENT d1d2...dn; DRAW/MODEL LOC d3d4
NOTE	The command Mirror Entity with or without modifier COPY followed by a colon will prompt ent. The entity, entities or groups of entities to be mirrored are then selected. The selection phase is terminated by issuing a semi-colon (;). The command will then prompt loc.
MODIFIERS	
COPY	Without SAMLAY this modifier specifies that the new entities are to be inserted as mirrored copies of some existing entities on the current active layer.
SAMLAY	If SAMLAY is used with COPY, the entity will be copied on the same layer as the existing entity.
EXAMPLES	#n#MIR ENT:MODEL ENT d1d2d3d4d5d6d7;MODEL LOC d8d9

END

4-53

d4

d2   d3   d5   d6

d1
d8   d7
d9
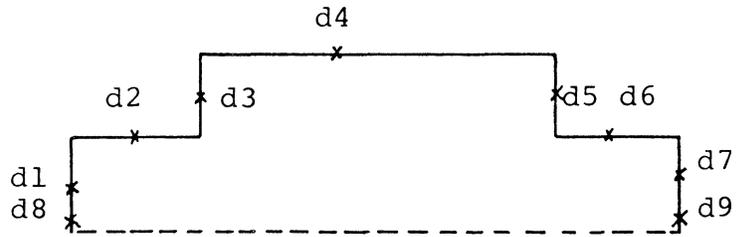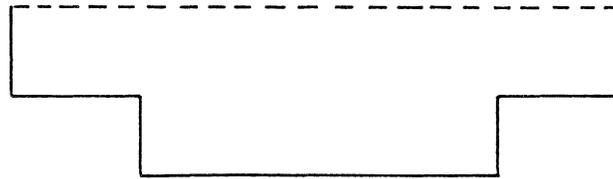
BEFORE

AFTER

#n#MIR ENT COPY:MODEL ENT d1;MODEL LOC d2d3

d1

d2

d3

BEFORE

AFTER

COMMAND	ROTATE ENTITY
DESCRIPTION	The Rotate Entity command can be used to rotate or rotate/copy in three-space up to 15,000 entities or group entities.
	The Rotate Entity command with the COPY modifier can be used to insert a rotated copy of the entities rather than the original entities.
	In either case, the rotated entities are placed on the same layer as the original entities. If COPY is specified, the user can control the insertion layer for the rotated entities by using layer modifiers.
	If a group is selected for rotation or rotation/copy, only the first 80 entities that compose the group are rotated.
	The rotated entities of a group are still grouped; however, the rotated/copied entities of a group are not themselves grouped.
	COPY AZ 60
SYNTAX	ROT ENT (modifiers): DRAW/MODEL ENT d1d2...dn DRAW/MODEL LOC d1d2...dn \S¦
MODIFIERS	
COPY	Used to rotate and copy the selected entities on the current active layer. Can be specified in either form of the command.
SAMLAY	If SAMLAY is used with copy, the entity will be copied on the same layer as the existing entity.
MODEL,VIEW	Reset the value of the model/view switch, which determines whether AX, AY, AZ and TX, TY, TZ refer to the model space axes or the view space axes.
AX,AY,AZ	Rotate by the given angle around the given axis in the positive sense as determined by the right hand rule.
	Example: AZ20 means, "Rotate 20 degrees counter-clockwise around the Z-axis."

PX,PY,PZ Point the indicated original axis in the direction indicated by the numerical argument. (1,2,3 is equivalent to X,Y,Z; -1,-2,-3 means in the negative X,Y,Z direction.)

Example: PX-2 means, "Point the X-axis in the negative Y direction."

MX,MY,MB Mirror the X coordinates, the Y coordinates, or both at once.

EXAMPLES

#n#ROT ENT⌃MODEL ENT d1; MODEL LOC d2d3d4

Copy (handwritten annotation)
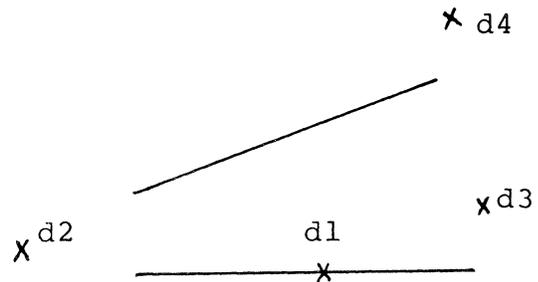
#n#ROT ENT COPY AZ72:MODEL ENT d1;MODEL LOC X0Y0N4


BEFORE   AFTER

|	
COMMAND	STRETCH ENTITY
DESCRIPTION	The Stretch Entity command stretches a protion of a part that is bordered by a window or polygon. Displacement is described by indicating two locations through digitizing or through explicit input. The distance and direction of the displacement is expressed by the distance and direction between the two locations. Parts may be stretched in any direction and made shorter, longer, wider, or narrower.
SYNTAX	STR ENT (modifiers): DRAW/MODEL LOC d1d2(d3...dn) DRAW/MODEL LOC d3d4
MODIFIERS	
WIN	The WIN modifier specifies that a rectangular window will be used to surround the entities being stretched (default).
PWIN	The PWIN modifier specifies a polygon window will be used.
LIN STG NLIN	Using any of these modifiers or a combination of these modifiers will permit only lines, strings or nodal lines to be stretched. Default is all three modifiers selected.
TRAN -TRAN	TRAN will cause entities within the window or polygon window to be translated. The modifier -TRAN will prevent lines, strings, and/or nodal entities completely within the window from being translated. Default is TRAN.
NOTE	Only string, line, and nodal line segments can be stretched; portions of subfigures, arcs, circles, fillets, etc. are not stretched but translated.
RESTRICTIONS	A maximum of 1024 entities may be stretched at one time.
EXAMPLES	#n#STR ENT WIN:MODEL LOC d1d2 MODEL LOC d3d4
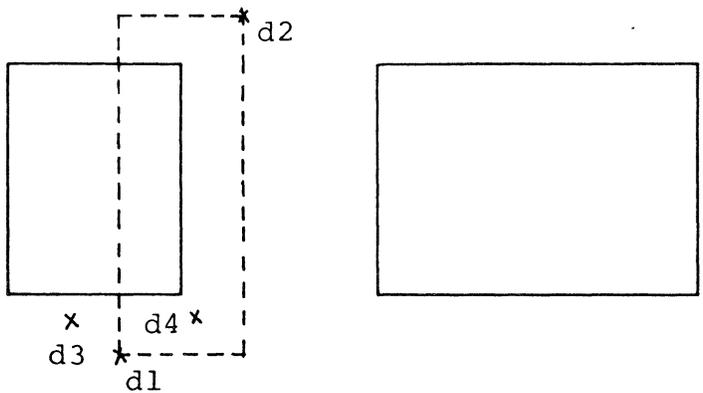
BEFORE   AFTER

#n#STR ENT WIN:MODEL LOC d1d2 MODEL LOC X2Y2IX-1.5
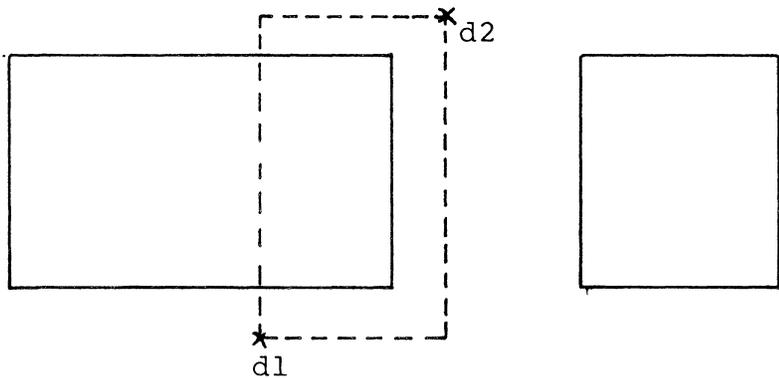
BEFORE   AFTER

#n#STR ENT PWIN:MODEL LOC d1d2d3d4d5;MODEL LOC d6d7
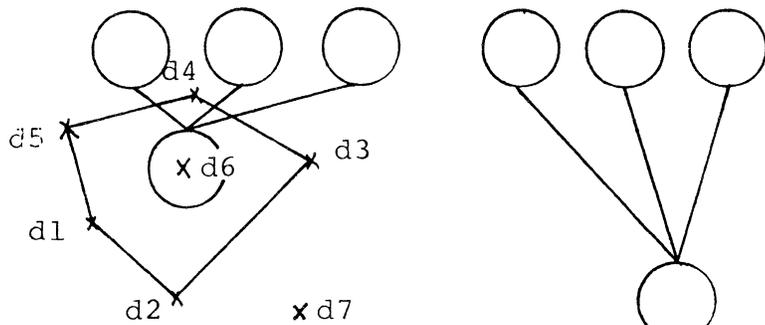
BEFORE   AFTER

COMMAND	TRANSLATE ENTITY
DESCRIPTION	The Translate Entity command moves any specified entity. The type of entity moved may be specified as a GETDATA modifier. When entities to be moved have been identified, a semicolon (;) followed by two coordinates, explicit or implied, indicate the begining and end of the translation vector.
SYNTAX	TRA ENT (modifiers): DRAW/MODEL ENT d1d2...dn ; DRAW/MODEL LOC d3d4
MODIFIERS	
NOCOPY	(Default) specifies entities not to be copied.
COPY	Specifies that the entity will be copied to the new location rather than moved.
SAMLAY	If SAMLAY is used with COPY, the entity will be copied on the same layer as the existing entity.
CAUTION	Interrupting this process may produce errors in the database.
EXAMPLES	#n#TRAN ENT:MODEL ENT d1; MODEL LOC d2d3

d1

d2 x d3 x

BEFORE   AFTER
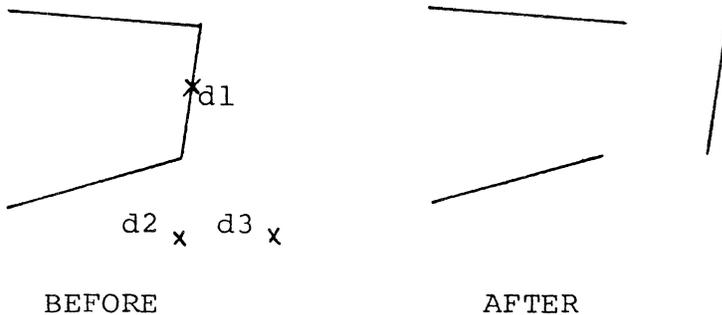
#n#TRAN ENT COPY:MODEL ENT d1;MODEL LOC d2d3

d1

d2 x   x d3

BEFORE   AFTER
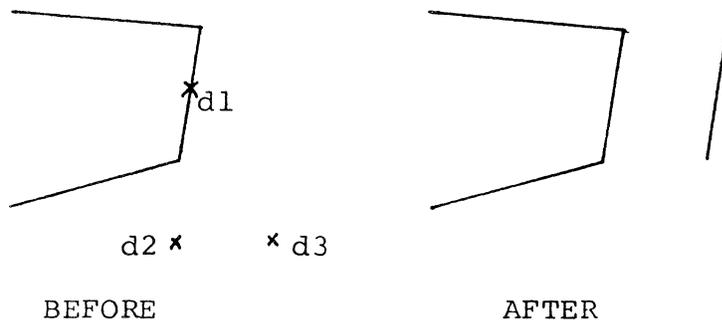
COMMAND	TRIM ENTITY
DESCRIPTION	The Trim Entity command stretches or deletes existing endpoints of the following entities:

|	
STRING	ELLIPSE
LINE	HYPERBOLA
CIRCLE	PARABOLA
ARC	SPLINE
FILLET	BSPLINE

SYNTAX

TRI ENT (modifiers): DRAW/MODEL ENT d1 DRAW/
MODEL LOC d2(d3)

MODIFIERS

LNGn
ILNGn

Are reserved for the entity line and restricted to a real number. Both specify the line should be trimmed to length n. Negative value will extend the line.

INTOF

Allows the trimming of an entity, in true three space, to its intersection with another entity.

EXAMPLES

#n#TRI ENT LNG4:MODEL ENT d1


BEFORE   AFTER
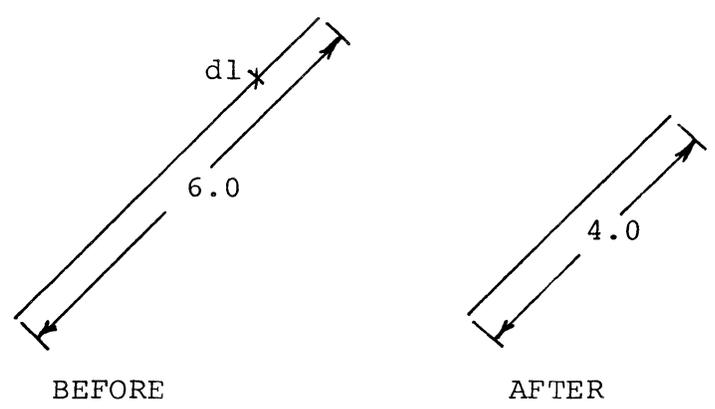
#n#TRI ENT:MODEL ENT CIR d1MODEL LOC d2d3

BEFORE   AFTER

#n#TRI ENT:MODEL ENT d1MODEL LOC d2

```
                                      d2
                          d1          ✗
        ──────────────────✗──         ⋮

        ─────────────────────────────
```

#n#TRI ENT INTOF:MODEL ENT d1d2

```
                                    ↙✗ d1

        ──────────────────────────────✗ d2
                                     /
                                    /
                                   /
                              BEFORE

        ─────────────────────────────
                                    /
                                   /
                                  /

                              AFTER
```

#n#TRI ENT INTOF:MODEL ENT d1d2d3

```
                      │             │
             d1       │             │
        ─────✗────────┼─────────────┼────
                      │             │
                    ✗ │d2         ✗ │ d3
                      │             │
                    BEFORE

                      │             │
                      │             │
                      │─────────────│
                      │             │

                    AFTER
```

COMMAND	TAG ENTITY
DESCRIPTION	The Tag Entity command attaches alphanumeric name tags, 1 to 6 characters in length limited to characters A-Z, 0-9, to specified entities.
SYNTAX	TAG ENT (modifiers) (CR)
MODIFIERS	
TYPE	Identifies a specific entity to tag. Entity types include:

ENT	CIR	ELP	BSPL
POI	ARC	PRB	
LIN	FIL	HYP	

|	
NOTE	ENT allows the digitizing of any entity.
APT	Apt language tags. Default.
SPLIT	Split language tags.
CMPCT	Compact-II language type.
ALL	Tags the entire database.
UNTAG	Tags all untagged entities.
NOCHK	Prevents the system from checking entities that have previously been tagged. Especially useful in execute files.
NOTE	TAG ENT requires at least one modifier and at most five.
ERROR MESSAGES	NON UNIQUE NAME RETRY
	If a tag name is specified more than once the system will display this message. Control will return to the modifier processor allowing the user to reenter a new set of modifiers.
	TYPE OK TO OVERWRITE
	When a previously tagged entity is tagged in another language and the modifier NOCHK has not been specified.

CONTRADICTORY INPUT, RETRY

If contradictory modifiers are entered (ENT
and LIN) this message will appear and control
will automatically return to the modifier
processor.

EXAMPLES

#n#ECHO TAG ON (CR)
#n#TAG ENT ALL SPLIT (CR)

Before   After
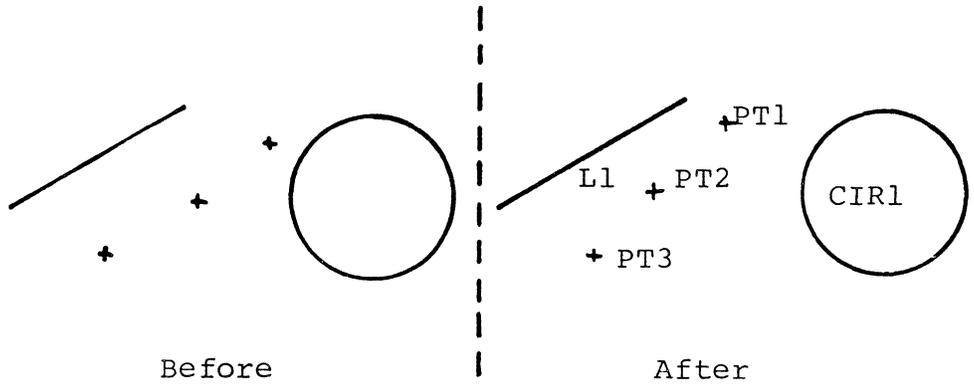
#n#TAG ENT ALL UNTAG (CR)

Before   After
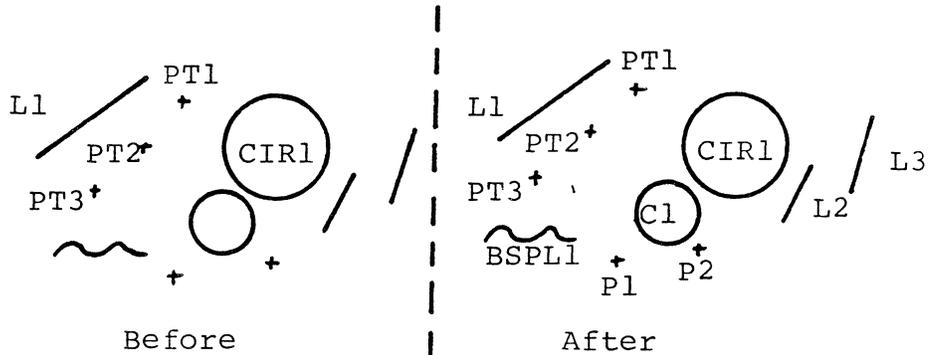
#n#TAG ENT CIR SMALL:MODEL ENT dl (CR)
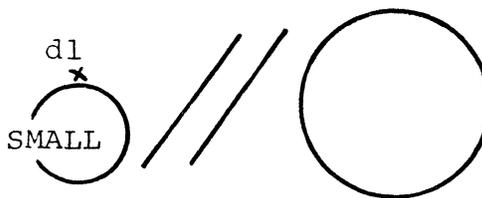
DATABASE VISUALIZATION COMMANDS

The Database Visualization commands are used to manipulate the way the model is displayed within the drawings and views. Database Visualization commands do not have a permanent effect upon the model database, but rather, are used to temporarily alter the way in which the model is perceived.

Functions controlled by these commands include:

* Blanking
* Fonting
* Scrolling
* Zooming
* Resetting the view to the original appearance

These commands include:

BLANK ENTITY
ECHO FRAME
ECHO GRID
ECHO LAYER
REPAINT
RESET VIEW
RESTORE IMAGE
SCROLL DRAWING
SCROLL VIEW
UNBLANK ENTITY
ZOOM DRAWING

COMMAND	BLANK ENTITY
DESCRIPTION	Blank Entity is a mode dependent command used to blank entities in either drawing or model mode. The entities are blanked in all views. The entities, blanked from the screen only, still remain in the part database.
SYNTAX	BLA ENT:DRAW/MODEL ENT d1d2...dn
EXAMPLES	#n#BLA ENT:MODEL ENT d1d2d3

TO RET. LINES
UNBLA ENT.)

d1

d2

d3
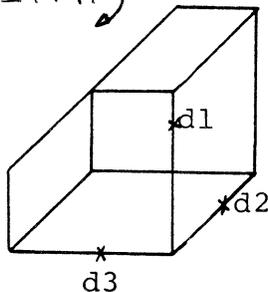
BEFORE
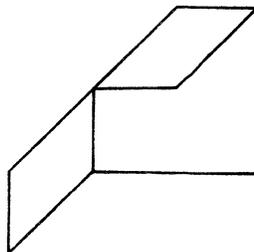
AFTER

INS. LINE APP FONT DASHED!

COMMAND	ECHO FRAME
DESCRIPTION	The Echo Frame command allows the user to visualize the clipping boundaries of specified views.
SYNTAX	ECH FRA (modifiers) (CR)
MODIFIERS	
OFF	Indicates that the display frames are to be turned off. If omitted the display frames are turned on.
EXAMPLE	#n#ECH FRA
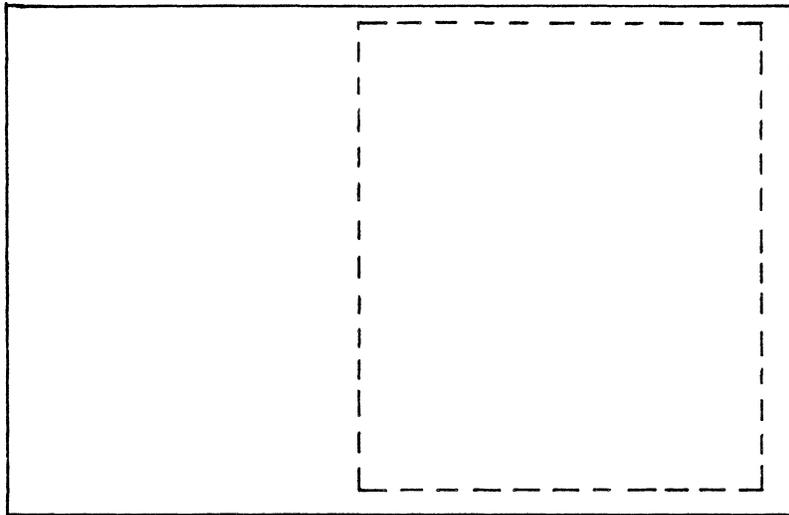
COMMAND	ECHO GRID
DESCRIPTION	The Echo Grid command controls the display of the grid for the active mode.
SYNTAX	ECH GRID (modifiers) (CR)
MODIFIERS	
ON	The grid displayed according to the current grid parameters. (Default)
OFF	The grid will no longer be displayed after the next screen repaint.
SNAP	Location digitizes are to snap to the nearest grid point. (Default with ON)
NOSN	Location digitizes are not to snap to the nearest grid point. (Default with OFF)
ERROR MESSAGES	SNAP IS INVALID WITH OFF, SNAP WILL BE IGNORED
	Displayed if the modifiers SNAP and OFF are entered.
	NO PART ACTIVE, COMMAND IGNORED
	Displayed when an attempt to ECHO GRID before a part is active.
EXAMPLE	#n#ECH GRI
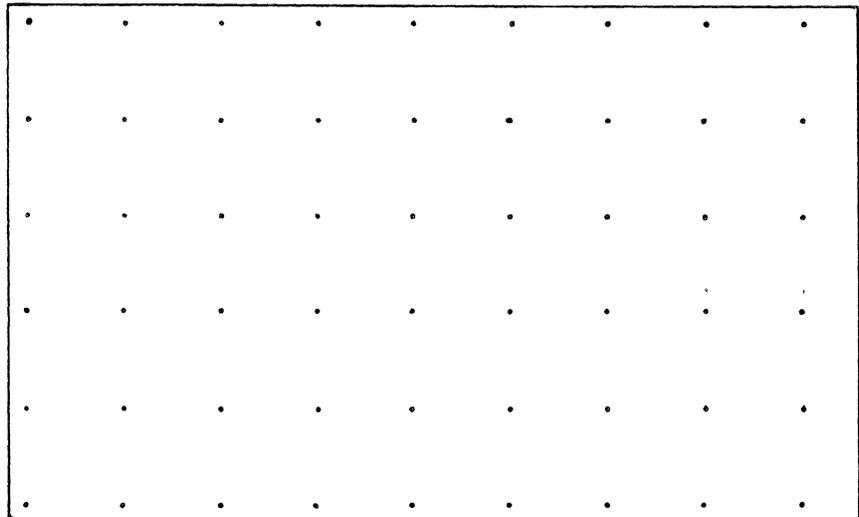
COMMAND	ECHO LAYER
DESCRIPTION	The Echo Layer command displays the current active layer or layers in an active part.
SYNTAX	ECH LAY (modifiers) (CR)
	ECH LAY (modifiers) :DRAW/MODEL VIEW d1d2...dn
	If a colon is used only views digitized are affected by the command.
MODIFIERS	
n	Layers to be displayed. Syntax can be in the form of a string separated by spaces (ECHO layer 0 4 7 9 11) or a from-to form (ECHO layer 1-3 47-50) or both (ECHO layer 0 4 20-30 41-48 68).
ALL	Echoes all layers.
INCLn	Echo layers listed after the INCL modifier plus currently visible layers.
EXCLn	Echo all layers except those listed after EXCL.
DRAW	Limits layer echoing to drawing entities.
EXAMPLES	#n#ECH LAY INCL23 (CR)

ACTIVE LAYER 15
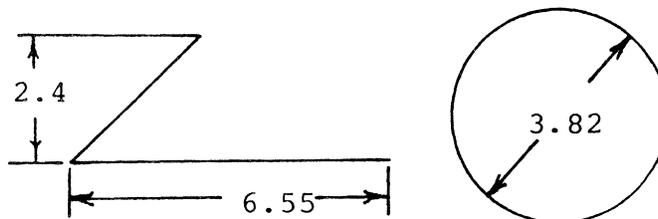
WITH LAYER 23 DIMENSIONS ECHOED

COMMAND

REPAINT

DESCRIPTION

The Repaint command causes the current graphics
image to redisplay. The command removes any
transitory changes to the image. For example,
it will redraw those parts of entities that
temporarily disappear from the screen when
an intersecting entity is deleted.

SYNTAX

REPA (CR)

EXAMPLES

#n#REPA (CR)
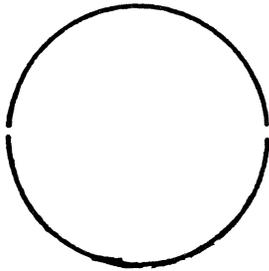
BEFORE
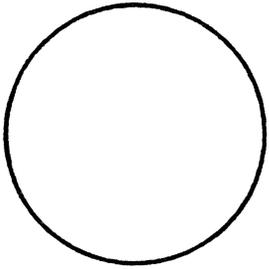
AFTER

COMMAND	RESET VIEW
DESCRIPTION	The Reset View command returns a view to its last permanent state.
SYNTAX	RES VIE (modifier) (CR)
	RES VIE: VIEW d1...dn
MODIFIERS	
ALL	All resets all views of the current active part. The colon resets views specified by the user.
EXAMPLES	#n#RESET VIE: VIEW d1
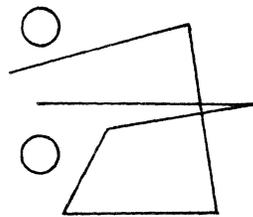
ORIGINAL VIEW
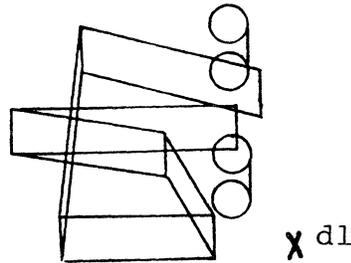
X d1

ALTERED VIEW
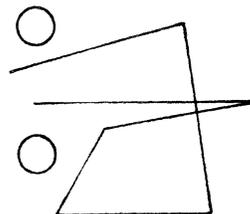
RESET VIEW

COMMAND	RESTORE IMAGE
DESCRIPTION	The Restore Image command redisplays an image saved by the Save Image command.
SYNTAX	RESTO IMA name (CR) Name is the name assigned to a particular saved image by the user. It can be only 20 characters long and contain no special characters.
EXAMPLES	#n#RESTO BLOCKS
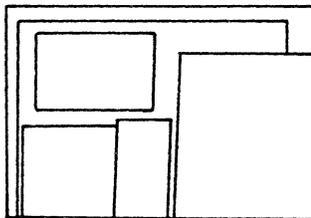
BLOCKS

COMMAND	SCROLL DRAWING
DESCRIPTION	The Scroll Drawing command permits the temporary alteration of a drawing by scrolling.
SYNTAX	SCR DRA (modifiers) (CR) SCR DRA:
MODIFIERS	
RIGHT LEFT UP DOWN	Direction modifiers used to display a drawing in their respective direction in half screen increments.
Rn	Specifies scrolling in terms of screen size.
Dn	Specifies scrolling in terms of drawing coordinates. The actual drawing image displacement along the screen would be a function of drawing size parameters and the current zoom state.
DATA INPUT	
:	The optional colon signals the system to center the scroll around the specified digitize point. If two or more digitizes are received the last digitize will track the one proceeding it.
EXAMPLES	#n#SCR DRA:MODEL LOC d1d2
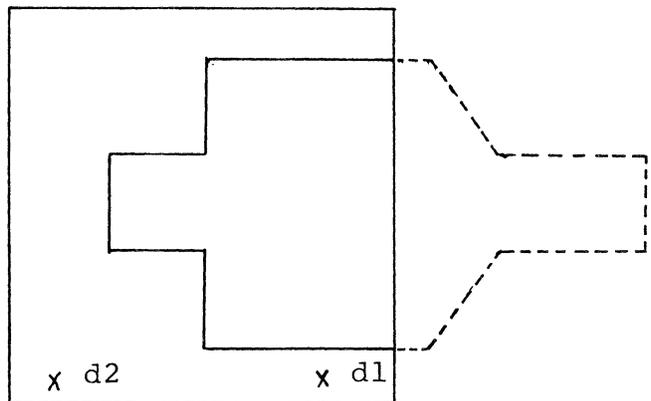
BEFORE
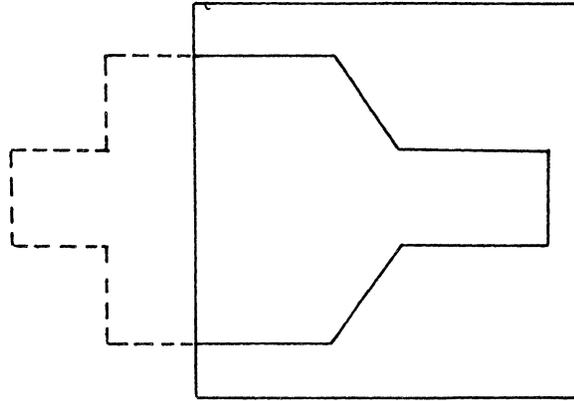
AFTER

COMMAND	SCROLL VIEW
DESCRIPTION	The Scroll View command temporarily moves model entities within a view to a fixed border thus moving model entities in and out of view.
SYNTAX	SCR VIE (modifiers); DRAW/MODEL VIEW d1 DRAW/ MODEL LOC d2d3
MODIFIERS	
RIGHT LEFT UP DOWN	Self explanatory modifiers that scroll the view in half screen increments.
R Rn	Are used in cojunction with the direction modifiers. Rn specifies various screen scrolling ratios, Rn specifies the distance on drawing unit.
EXAMPLES	#n#SCR VIE:VIEW d1; MODEL LOC d2d3
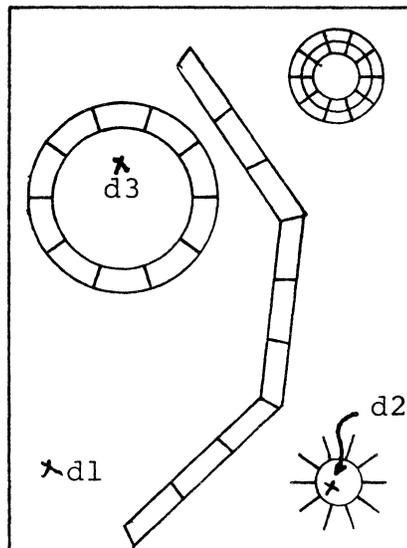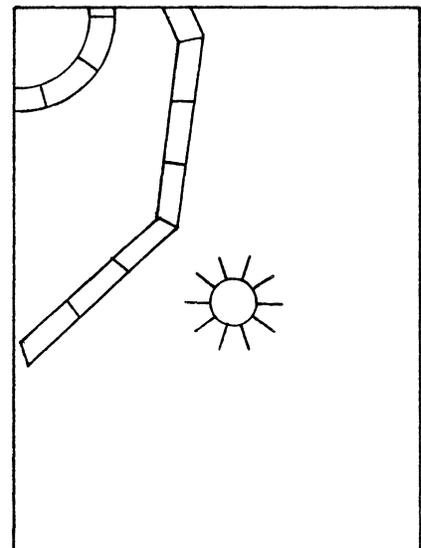
BEFORE   AFTER

COMMAND	UNBLANK ENTITY
DESCRIPTION	The Unblank Entity command redisplays all entities blanked by the Blank Entity command.
SYNTAX	UNB ENT (CR)
NOTE	Unblanks all entities that were blanked at any time in the part. No selection is possible.
EXAMPLES	#n#UNB ENT (CR)
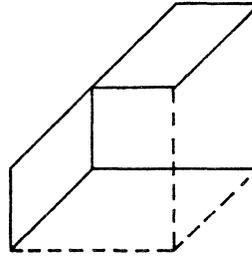
BEFORE
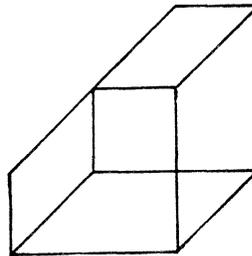
AFTER

COMMAND	ZOOM DRAWING
DESCRIPTION	The Zoom Drawing command permits the operator to temporarily alter a drawing by zooming. A digitize selects a point that remains fixed during the zoom operation. If no location is given the focal point defaults to the center of the screen.
SYNTAX	ZOO DRA (modifiers): DRAW/MODEL LOC d1d2
MODIFIERS	
IN	The drawing is redisplayed larger on the screen by a factor twice its present size.
OUT	The drawing is redisplayed smaller on the screen by a factor one half its present size.
An	The drawing is redisplayed on the screen by a factor of n its true size. For n = 1.0, drawing images will appear true size on the CRT display.
Rn	The drawing is redisplayed on the screen by a factor of n from its present size.
ALL	The drawing is both zoomed and scrolled so as to exactly fit within the bounds of the CRT screen.
WIN	The drawing is both zoomed znd scrolled in order that the window defined by the two opposite corner digitizes exactly fills the viewing screen.
RESTRICTIONS	The Zoom Drawing command is valid in drawing and model modes. The system will reject a colon after the modifier ALL.
EXAMPLES	#n#ZOO DRA WIN:MODEL LOC d1d2
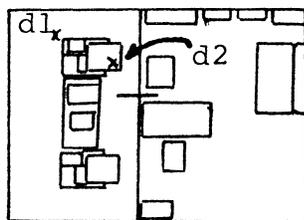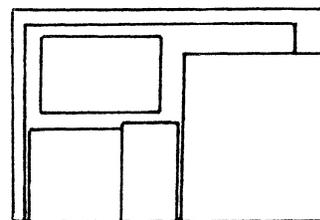
BEFORE   AFTER

DATABASE CONSTRUCTION PROCESSES COMMANDS

Database Construction Processes commands provide more sophisticated ways of constructing or editing the model database. In general, these commands invlove processes which require some database setup before the execution of the command. Often, these commands require much more detailed data input specifications than do the database construction commands; and some of these commands are useful only for specific applications.

These commands include:

    EDIT TEXT

COMMAND	EDIT TEXT
DESCRIPTION	The Edit Text command provides a means of modifying text or nodal text entities in drawings. The command applies to the editing of single or multiple instances of text (or nodal texr) entities.
	All text lines from all selected entities are placed in a large buffer and numbered sequentially. The system will prompt with , which signals the user to enter an editing command. After each line of edit specification, the user types (CR), which causes the editing command to be executed and actually modifies the line or text buffer. The system prompts and awaits the next edit command. The original entities retain their original content until the file command is entered.

SYNTAX	EDI TEXT (modifiers) (CR)
	    Allows editing of all text on specified
	    layers.

	EDI TEXT (modifiers):MODEL ENT d1d2...dn (CR) Allows editing of all digitized text.
MODIFIERS	
LAY n,m-p,q	Specifies the layers or ranges of layers containing text to be edited. Default is visible layers.
CASE	Specifies that the edit is to apply to both upper and lower case letters even though an upper-case-only keyboard is being used. In-stream modifiers will appear in the text string to be edited.
ALL	All text may be edited.
EDITING COMMANDS	The commands listed below may be used after the edit prompt " ".

I(n)	Insert after line n
Dn(-m)	Delete lines n to m
T(n-m)	Type lines n to m
Rn(-m)	Replace lines n to m
L(n-m),XXX	Locate and type lines containing text XXX
S(n-m),XXX,YYY	Substitute YYY for XXX in lines n to m

```
                      En                    Edit line n
                      Q                     Quit (and ignore all edits)
                      F                     File
```

CHARACTER
CHANGE
KEYS

The characters listed below serve to add or
delete characters in a text string.

Cont

  [- A bracket inserts a character just
     before the character it is under.
  ^ - A carret deletes the character it is
     under.
  \ - A backslash deletes the character it
     is under and shifts all remaining
     text to the left by one space.

EXAMPLE

The following are text entities that need to
be edited in order for the drawing to be
correct.

```
     ALL MATEIAL
     CAST IRN

EDI TEXT:MODEL ENT dl (CR)
>E 1 (CR)
ALL MATEIAL
        [R (CR)
ALL MATERIAL (CR)
>E 2 (CR)
CAST IRN
        [O (CR)
CAST IRON (CR)
>F (CR)

     ALL MATERIAL CAST
     CAST IRON

EDI TEXT:MODEL ENT dl (CR)
>E 1 (CR)
ALL MATERIAL CAST
              ^^^^ (CR)
ALL MATERIAL (CR)
>F (CR)

     ALL MATERIAL CAST CAST IRON

EDI TEXT:MODEL ENT dl (CR)
>E 1 (CR)
ALL MATERIAL CAST CAST IRON
                  \\\\ (CR)
ALL MATERIAL CAST IRON (CR)
>F (CR)
```

INTERACTION
MANAGEMENT
COMMANDS

Interaction Management commands are used to control the CADDS 4 interactive environment. Functions include default parameter and operational mode selection and function key management.

These commands include:

RESTORE KEY
SAVE IMAGE
SELECT GRID

COMMAND	RESTORE KEY
DESCRIPTION	The Restore Key command restores a Saved Key File. All current definitions are lost. (A Saved Key File is created with the Save Key command.)
SYNTAX	RESTO KEY (CR) KEYFILE name TRG. 4

The system requests the name under which the key definitions will be found. The system starts the line by typing the first part of the name, KEYFILE, which is the name of the base catalog under which all key files are stored. The user supplies the rest of the name (the unique part).

ERROR MESSAGES

    RSTKEY: FILE NOT FOUND

The file name specified does not match any existing file.

    RSTKEY: INVALID FUNCTION KEY FILE

The specified file was found, but did not contain function key definitions.

    RSTKEY: TOO MANY KEYS

The file is too large to fit into the allowed space for function keys.

EXAMPLE

#n#RESTO KEY (CR)
KEYFILE TRG 4 (CR)

COMMAND	SAVE IMAGE
DESCRIPTION	The Save Image command stores the current active drawings zoom and scroll state. All saved images will be filed with the part.
SYNTAX	SAV IMA name (CR) Name is an assigned name of a particular image that is less than 20 characters long and contains no special characters.
EXAMPLES	#n#SAV BLOCKS
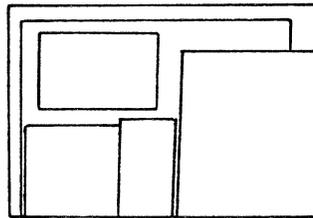
BLOCKS

COMMAND	SELECT LAYER
DESCRIPTION	The Select Layer command selects a construction layer for an active part. Entities subsequently digitized will reside on the selected layer.
SYNTAX	SEL LAY n (CR)
	n must be an integer between 0 - 254 preceded by a space. Default is 0.
EXAMPLE	#n#SEL LAY 10 (CR)

COMMAND	SELECT GRID
DESCRIPTION	The Select Grid command is used to specify desired grid parameters. Its mode is determined by the mode of the current active part. If -o part is active the grid defaults to model mode. If new grid parameters are specified, the grid must be reechoed to reflect the new values.
SYNTAX	SEL GRI (modifiers)(CR)
MODIFIERS	
MODEL	Specifies parameters for a model grid.
DRAW	Specifies parameters for a drawing grid.
ORIG	The operator will digitize a grid origin at the conclusion of the command.
REC	A rectangular grid is selected.
RAD	A radial grid is selected.
RECTANGULAR MODIFIERS	
DYn	Specifies spacing between grid points on the Y-axis.
DXn	Specifies spacing between grid points on the X-axis.
DGn	Specifies uniform spacing in X and Y. "n" is the spacing between grid points in database units.
ROTATION MODIFIERS	
DSn	"n" is the radial distance between grid points.
DAn	"n" is the number of degrees per sector. If it is not an even multiple, the last sector (counter-clockwise from the horizontal) will be short.

AXn AYn AZn ANGn	These modifiers define a 3-dimensional rotation for a 2D grid. Their values are in degrees and define the extent and direction of the rotation. The rotation is from the model axis or from the axis of a specified view. The rotation will be expressed as a 3 x 3 orientation matrix parameter. These modifiers are not commutative; their order affects the final orientation. (ANG is equivalent to AZ).
MODEL	The rotations are from the model axis. Model is the default.
ORG	Specifies grid origin is to be digitized.
VIEW	The rotations are from a view axis. The user is immediately prompted to digitize or name a view.
EXAMPLES	#n#SEL GRI AZ45 #n#ECH GRI
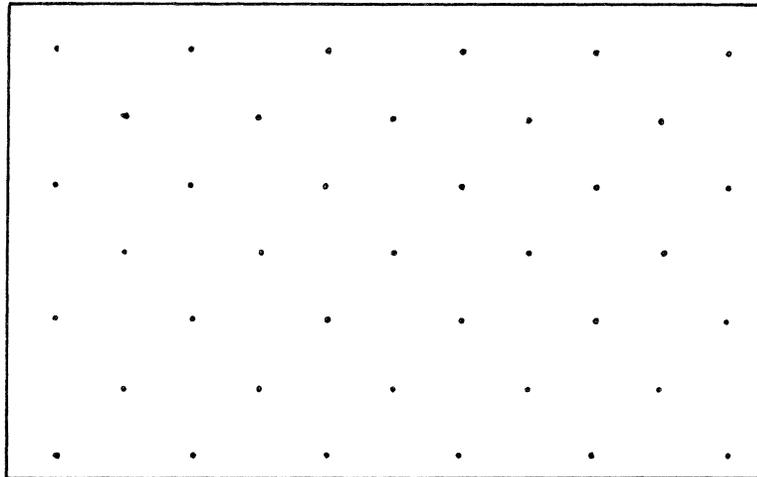
ECHO GRID DEF

ARROWHEADS	The following list shows the various types of arrowheads that are available to the operator.
STANDARD	The CV standard unfilled triangular arrowhead (default).
	CV standard ⟶▷
FILLED	Specifies filled arrowheads.
	Filled ⟶▶
OPEN	Specifies open arrowheads.
	Open ⟶➤
DOT	Specifies dimension line ends with a dot.
	Dot ⟶●
NONE	Specifies no arrowheads appear.
	None ⟶
DIAMETERn	Specifies the diameter of the dot. Default is .125cm/.05in.
LENGTHn	Specifies the length of the arrowhead. Default is .4cn/.15in.
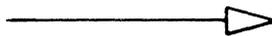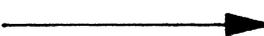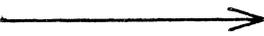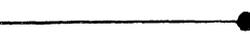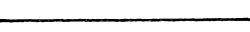
(default is .15)

RATIOn	Specifies the ratio of the arrowhead length to the base width. Default is 3.

$n = a/b$

(default is 3)
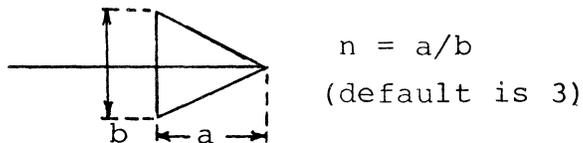
NOTE	All default values for ARROWHEAD parameters listed here are standard defaults. However, the user may define other global defaults via the Select Dimension command before inserting any dimensions.

COMMAND

DESCRIPTION

SELECT DIMENSION

The Select Dimension command is used to define
parameters that control subsequent dimension
creation. In most cases, the individual dimension
insertion commands have corresponding modifiers
which can be used to override the parameters set
by the Select Dimension command. In addition
to dimensions, labels, feature control symbols,
and flags are also affected. Defaults come
from the default parameter file which is user
definable.

If the Select Dimension command is used while
not in an active part, the specified parameters
will be the default parameters for any part
entered during the CADDS session. If used while
in an activated part, the specified parameters
will be in effect for that part only. When CADDS
is re-entered all parameters are intialized to
the standard default state.

The customary dimensioning units used are desig-
nated as "primary". If an additional unit system
is used, it is called "secondary". The Select
Dimension command is used to specify the primary
and secondary units for dimensioning a part.
These need not be the same as the units used in
creating the model. For example, model units
can be millimeters, while the primary dimensions
are in centimeters and the secondary dimensions
are in inches.

The Select Dimension command is used to designate
on which layer the primary dimensions will appear
and on which layer the secondary dimensions will
appear. When both primary and secondary dimen-
sioning is activated, two dimensions (one primary
and one secondary) will be created simultaneously
on separate layers by any single dimension
insertion command.

Dual dimensioning is another alternative. A dual
dimension contains both primary and secondary
dimension values together within a single dimen-
sion entity. As with primary and secondary
dimensioning, dual dimensions will appear on a
designated layer. In fact, it is possible to
have all three modes: primary, secondary, and
dual, or any combination ot the three, active
simultaneously.

SYNTAX	SEL DIM (modifiers) (CR)
MODIFIERS	The Select Dimension command includes numerous modifiers for versatility. For ease of using these modifiers, they are grouped in a hierarchical structure of associated modifiers. Four levels of modifiers are listed: first-level, second-level, third-level, and fourth-level. The corresponding level is indicated in each modifier description below. Second-level modifiers are dependent upon the associated first-level modifiers; third-level modifiers are dependent upon the associated second-level modifier; and fourth-level modifiers are dependent upon the associated third-levle modifiers. For example, if the first-level ARROWHEAD modifier is selected, the second-level OPEN modifier may be used. Similarly, if the first-level TEXT modifier is used, the second-level FORMAT modifier may be used, then the third-level FRACTION modifier may be used, and finally, the fourth-level SMALL modifier may be used. A table summarizing all associated modifiers follows the descriptions below.
ARROWHEAD	Controls the arrowhead parameters. First level.
STANDARD	The Computervision standard unfilled triangular arrowhead type is used. Default. Second level.
FILLED	Specifies a filled arrowhead. Second level.
OPEN	Specifies an open arrowhead. Second level.
DOT	Specifies a dot. Second level.
NONE	Specifies no arrowhead. Second level.
LENGTHn	Specifies arrowhead length. Default is .4cm/ .15 in. Second level.
RATIOn	Specifies arrowhead length to width ratio. Default is 3. Second level.
DIAMETERn	Specifies dot diameter. Default is .125cm/ .05 in. Second level.
LAYER	Specifies layer for dimensions. Default is current layer. First level.

PRIMARYn	Specifies primary dimensions appear on layer n. Default. Second level.
SECONDARYn	Specifies secondary dimensions appear on layer n. Second level.
DUALn	Specifies dual dimensions appear on layer n. Second level.
NOTE	For all of the above, if n is -1, the current layer will be used.
PRECISION	Controls the number of decimal places to the right of the decimal point for decimal format or the denominator value (1,2,4,8,...64) for binary fraction format for the dimension value. First level.
PRIMARYn	Specifies primary precision will be n. Second level.
SECONDARYn	Specifies secondary precision will be n. Second level.
BOTHn	Specifies both primary and secondary precision will be n. Second level.
ANGULAR	Specifies angular precision will be n. Second level.
TEXT	Controls text characteristics. First level.
FIXED	The dimansion value is automatically calculated, but thereafter is treated as though the user had keyed it in. Regeneration will not change the dimension value. Second level.
UNFIXED	The dimension value will not be fixed. Default. Second level.
FORMAT	Controls the format used for the dimension value. Second level.
DECIMAL	Decimal fractions will be used. Default. Third level.
FEET & INCHES	Feet-inch format will be used, i.e., 1'-10". The fraction modifier should be used to indicate the desired fraction format. Third level.

FRACTION	Fractional format will be used. The size of the denominator is controlled by the PRECISION modifier. Third level.
SMALL	Specifies a fraction whose total height is equal to the normal character height. A horizontal fraction bar is used. Fourth level.
MEDIUM	Specifies a fraction whose total height is equal to the normal character height. A slash separates numerator and denominator. Fourth level.
LARGE	Specifies a fraction whose numerator and denominator are both at normal character height. A horizontal fraction bar is used. Fourth level.
OFFSET	Specifies a fraction whose numerator and denominator are offset horizontally and are separated by a slash. The numerator, denominator, and slash are all at normal character height. Fourth level.
POINT	Specifies the decimal point. Third level.
PERIOD	Use a period for the decimal point. Fourth level.
COMMA	Uses a comma for the decimal point. Fourth level.
LOCATION	Controls text location. Second level.
AUTOCENTER	Text will be automatically centered. Third level.
MANUAL	Text will not be automatically centered. Third level.
HORIZONTAL	Specifies that for solid angular or linear dimensions the text will be horizontal. Second level.
TANGENT	Specifies that for solid angular or linear dimensions the text will be tangent to the dimension line. Second level.
TOLERANCE	Controls the tolerance specification. First level.
PRIMARY	Controls tolerance characteristics of the primary dimension. Second level.
POSITIVEn	Positive tolerance value will be n. Third level.

NEGATIVEn	Negative tolerance will be n. (Note: n should be a positive number.) Third level.
BOTHn	Both positive and negative tolerance values are n. Third level.
SECONDARY	Controls tolerance characteristics of the secondary dimension. Second level.
POSITIVEn	Positive tolerance will be n. Third level.
NEGATIVEn	Negative tolerance will be n. (Note: n should be a positive number.) Third level.
BOTHn	Both positive and negative tolerance values are n. Third level.
POSITIVEn	Specifies that the primary positive tolerance for a dimension is n. Second level.
NEGATIVEn	Specifies that the primary negative tolerance for a dimension is n. Second level.
BOTHn	Specifies that both primary positive and negative tolerance values for a dimension are n. Second level.
PRECISION	Controls the number of decimal places for tolerance values. Second level.
PRIMARYn	Primary dimension tolerances have a precision of n. Third level.
SECONDARYn	Secondary dimension tolerances have a precision of n. Third level.
BOTHn	Both primary and secondary dimension tolerances have a precision of n. Third level.
ANGULARn	Angular tolerances will have a precision of n. Third level.
NOMINAL	Tolerance precision will be set equal to nominal value precision. Third level.
TYPE	Controls the tolerancing method. Second level.
INCREMENTAL	Incremantal tolerancing is used. Default. Third level.

DASHED	Dashed tolerancing is used. Third level.
LIMIT	Limit tolerancing is used. Third level.
ROUNDED	Limit tolerancing with rounding is used. Third level.
ANGULAR	Controls the number of decimal places for Angular Dimension tolerance values. Second level.
POSITIVEn	Positive tolerance value will be n. Third level.
NEGATIVEn	Negative tolerance will be n. (Note: n should be a positive number.) Third level.
BOTHn	Both positive and negative tolerance values are n. Third level.
ON	Turns tolerancing on. Second level.
OFF	Turns tolerancing off.Default. Second level.
UNITS	Controls the units used for dimensioning. These may be different than both model and drawing units. First level.
ANGULAR	Controls angular units. Second level.
DEGREES	Angular Dimension values will be given in degrees and fractions of degrees. Default. Third level.
MINUTES	Angular Dimnesion values will be given in degrees, minutes, and fractions of minutes. Third level.
SECONDS	Angular Dimension values will be given in degrees, minutes, seconds, and fractions of seconds. Third level.
LINEAR	Controls linear units. Second level.
PRIMARY	Controls units for primary dimensions. Third level.
INCHES	Measurement in inches. Default (English). Fourth level.
FEET	Measurement is in feet. Fourth level
MILES	Measurement is in miles. Fourth level.
MILLIMETERS	Measurement is in milimeters. Fourth level.
CENTIMETERS	Measurement is in centimeters. Default (metric) Fourth level.

METERS	Measurement is in meters. Fourth level.
KILOMETERS	Measurement is in kilometers. Fourth level.
SECONDARY	Controls units for secondary dimensions. Third level.
NOTE	Fourth level modifiers are the same as those for Primary
BOTH	Controls units for both primary and secondary dimensions. Third level.
NOTE	Fourth level modifiers are the same as those for Primary.
REGENERATION	Controls the timing of dimension regeneration. First level.
DEFERRED	Regeneration will be deferred until a Regenerate Dimension command is used. Second level.
IMMEDIATE	All related dimensions will automatically be regenerated immediately after any geometry modification. Default. Second level.
MODEL	Dimensions to be model entities. They appear in all views and are subject to view clipping. Only model entities can be dimensioned. First level.
DRAW	Dimensions to be drawing entities. Either model or drawing entities can be dimensioned. Default. First level.
ASSOCIATIVE	Dimensions will be associated with the entities they dimension. First level.
NONASSOCIATIVE	Dimensions are not associated with the entities they dimension. First level.
STANDARD	Controls drafting standard used. First level.
ISO	International Organization for Standardization standards to be used. Text appears above the dimension line. Standard arrowheads. Default (metric). Second level.
JIS	Japanese Industrial Standards to be used. Text appears above the dimension line. Open arrowheads. Second level.

RIGHT	The bracketed dimension value to be to the right of the primary. Third level.
LEFT	The bracketed dimension value to be to the left of the primary. Third level.
ON	Turns dual dimensioning on. If the POSITION or BRACKET modifiers have not been specified, POSITION ABOVE is used. Second level.
OFF	Turns dual dimensioning off. Default. Second level.
CHECK	Checks are made to insure that a valid dimension with a minimum dimension line, extension line, and leader lengths are created. Adjustments will be made to the text location automatically as necessary. Default. First level.
NOCHECK	No check is made for dimension validity. This method is normally only used for Linear Dimensions inserted in chains, where the text is moved by the Edit Dimension command under user control, if necessary. First level.
LEADINGO	A leading zero is used before the decimal place of numbers less than one. Applies to English system only; SI numbers always use leading zeroes. First level.
NOLEADING	No leading zero will be used. Default. First level.
PRIMARY	Controls primary dimensioning status. First level.
ON	Turns primary dimensioning on. Default. Second level.
OFF	Turns primary dimensioning off. Second level.
SECONDARY	Controls secondary dimensioning status. First level.
ON	Turns secondary dimensioning on. Second level.
OFF	Turns secondary dimensioning off. Default. Second level.
RESTRICTIONS	None

ANSI	American National Standards Institute standards to be used. Text appears between dimension lines. Standard arrowheads. Default (English). Second level.
STRUCTURAL	American structural practice to be followed. Text appears above dimension line. Standard arrowheads. Second level.
SOLID	Solid dimension lines to be used. Text is placed above the line. Default (metric). First level.
NONSOLID	Nonsolid dimension lines to be used. Text is placed between the dimension lines. Default (English). First level.
PROPERTY	Specifies that the dimension value to be computed from coordinate properties associated with the dimension entities. First level.
NOPROPERTY	The dimension value to be based on coordinate values. Default. First level.
DUAL	Controls dual dimensioning specification. First level.
POSITION	The position method is to be used. Default. Second level.
ABOVE	The primary dimension value to be above the secondary. Default. Third level.
BELOW	The primary dimension value to be below the secondary. Third level.
RIGHT	The primary dimension value to be to the right of the secondary. Third level.
LEFT	The primary dimension value to be to the left of the secondary. Third level.
BRACKET	The bracket method is to be used. The secondary dimension value is always in brackets. Second level.
ABOVE	The bracketed dimension value to be above the primary. Third level.
BELOW	The bracketed dimension value to be below the primary. Third level.

**PART
VERIFICATION
COMMANDS**

The Part Verification commands provide the
user with the ability to interrogate the
system about the current state of graphics
and insertion paramenters or to check the
definition of database entities.

Functions include:

* Listing of selected parameters and
  valid parameter options.
* Measuring distances and angles.
* Verifying entity coordinate location,
  parameters and graphics.

These commands include:

LIST DRAWING
LIST LAYER
LIST PART
MEASURE ANGLE
MEASURE AREA
MEASURE DISTANCE
MEASURE LENGTH
VERIFY ENTITY

COMMAND	LIST DRAWING
DESCRIPTION	The List Drawing command is used to display various status information about one or more drawings in the active part.
SYNTAX	LIS DRA (modifiers) (CR)
MODIFIERS	
STATUS	Output all inclusive report on each drawing.
VIEWS	List only view names associated with each drawing.
SIZE	List only drawing size units.
ZOOM	List only current absolute zoom ratio (applicable to the acive drawing only.)
SCROLL	List only current absolute scroll position (active drawing only). This returns the point in the drawing (in drawing coordinates) which is currently located in the center of the screen.
ACTIVE	(Default) Give report on the active drawing only.
drawing name	Specifies name of other drawing in active part to report on.
ALL	Give a report on all drawings in the active part. (Any combination of the above modifiers may be used.)
EXAMPLES	#n#LIS DRA STATUS

```
**DRAWING    :   CV.KEY.SWITCH.MATRIX
  CREATED    :   11-04-82,18:22MODIFIED 11-12-82,15:11
  WIDTH      :   11.000     HEIGHT 8.500      UN: IN
  ZOOM       :   1.0000
  SCROLL     :   X=5.150    Y=-3.207
  VIEW       :   TOP
  VIEW       :   RIGHT SIDE
  VIEW       :   AUX 1
  VIEW       :   AUX 2
```

COMMAND	LIST LAYER
DESCRIPTION	The List Layer command lists the layers of the current active part.
SYNTAX	LIS LAY (modifiers) (:) or (CR)
NOTE	The colon is optional for the modifier ALL. Only views digitized are affected by the command.
MODIFIERS	
ALL	Lists all visible layers
ACT	Lists active construction layer.
DRAW	Lists all visible active drawing layers.
MODEL	Lists all visible active model layers.
EXAMPLES	#n#LIS LAY ALL

```
ACTIVE CONSTRUCTION LAYER     25

DRAWING: 1                 0-254

VIEW: 1                    0,2-254

VIEW: 2                    0-1,254
```

COMMAND	LIST PART

DESCRIPTION

The List Part command allows the operator either to list parts currently stored on disk or to list information about an active part. This command is valid in Model mode and Drawing mode and can be invoked either within an active part or outside of an active part. Along with each listed part is its creation date and time and last access date. Parts can be listed either by specifying a catalog or by having all parts listed.

SYNTAX

LIS PAR (modifiers) (CR)

MODIFIERS

CATALOG name

Catalog in which parts reside.

STATUS

Specifies the following information about the current part to be printed: part precision, dimension, model data base units, and the part extents. This modifier overrides the other two modifiers and can be used only within an active part.

ERROR MESSAGES

FILE NOT FOUND

When specifying a catalog name that does not exist.

NO PART ACTIVE

This message will print when the modifier STATUS is used outside of a CADDS part.

EXAMPLES

#n#LIS PAR CATALOG TRG.XXX

	CREATION		
PART NAME	DATE	TIME	ACCESS DATE
TRG.XXX.EX1	11-4-82	15:23:45	11-22-82
TRG.XXX.EX2	11-5-82	16:34:21	11- 6-82
TRG.XXX.EX3	11-5-82	14:02:55	11-28-82
TRG.XXX.EX4.LINE	11-7-82	17:15:27	11-22-82
TRG.XXX.EX.POINT.1	11-9-82	19:25:44	12- 4-82

COMMAND	MEASURE ANGLE
DESCRIPTION	The Measure Angle command measures the inclination and included angles of two intersecting lines.
SYNTAX	MEA ANG (modifiers): DRAW/MODEL ENT d1d2
MODIFIERS	
CPL	Computes the angles as they appear projected into the active construction plane.
MODEL	Angles are computed of the model in the space.
HOR	Computes the included angle of specified lines.
EXAMPLES	#n#MEA ANG:MODEL ENT d1d2

```
#n#MEA ANG:MODEL ENT d1d2
IN MODEL SPACE
     ACUTE ANGLE = 30.0
     OBTUSE ANGLE = 150.0

MODEL ENT d3d4

IN MODEL SPACE
     ACUTE ANGLE = 30.0
     OBTUSE ANGLE = 150.0

MODEL ENT (CR)
```
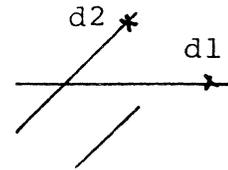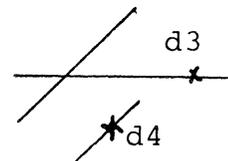
```
#n#MEA ANG:MODEL ENT POI d1d2d3

IN MODEL SPACE
     ACUTE ANGLE = 30.0
     OBTUSE ANGLE = 150.0

MODEL ENT (CR)
```

|	
COMMAND	MEASURE AREA
DESCRIPTION	The Measure Area command measures the area inside a closed polygon formed by lines, nodal lines, line strings and arcs, or an area defined by a set of points. It also calculates the perimeter enclosing an individual polygon, and calculates the averaged coordinate X and Y.
	The sides defining the polygon do not have to have their ends at the intersection points. This solves the problem of measuring the area inside a tic-tac-toe shape.
	The difference of the sum of two areas may be calculated by the use of the delimiters A or S.
SYNTAX	MEA ARE (modifiers): DRAW/MODEL ENT d1d2...dn
MODIFIERS	
ADD	Indicates that the result for the next polygon is added to the currently stored value.
SUB	Indicates that the result for the next polygon is subtracted from the currently stored value.
NOTE	The currently stored value will be initialized to 0 upon entering the command. If the user wishes to use the above modifiers, he should type a period to reenter the modifier processor and specify one of the modifiers. This mode will remain in effect until modifers are re-specified.
COOR	Indicates that the polygon is defined by a series of points, digitized or typed coordinates.
RESTRICTIONS	1. B-splines, or conic sections are not allowed as entities.
	2. The total number of sides in one operation cannot be more than 92. In other words, the total intersection points cannot be greater than 92.
	3. Each polygon should be closed. Each two

consecutive entities should have one inter-
section; otherwise, the error message:
POLYGON IS NOT CLOSED will be printed and
the calculations for that polygon will be
ignored.

4. An area can be defined by one single entity
   if that entity is a circle or a string,
   however, a string cannot cross itself to
   define two or more areas. This will cause
   an error.

EXAMPLES

#n#MEA ARE:MODEL ENT d1d2d3

AREA = 4.26753
ADD:MODEL ENT d4d5
TOTAL AREA=7.93507
SUB:MODEL ENT d6d7d8d9
TOTAL AREA=3.50526
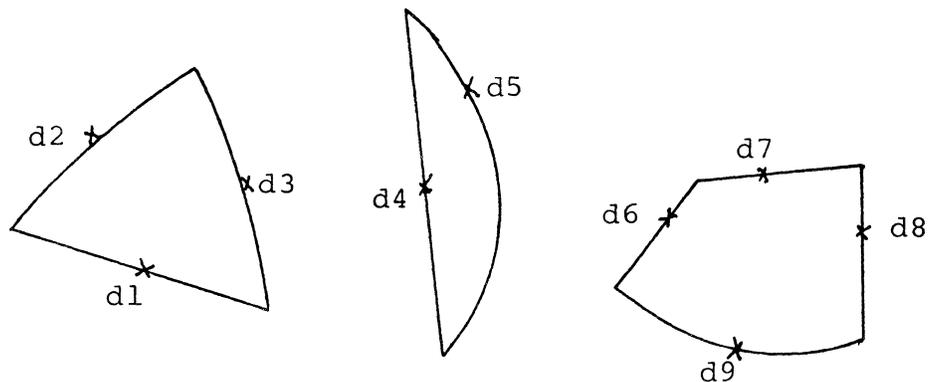
COMMAND	MEASURE DISTANCE
DESCRIPTION	The Measure Distance command allows the user to measure the shortest distance between two lines, a line and a point, or two points.
SYNTAX	MEA DIS (modifiers): DRAW/MODEL ENT d1d2
MODIFIERS	
MODEL	Measures the true distance in the 3-D model space. Default.
CPL	Measures the distance with the entities projected on the active construction plane.
ERROR MESSAGES	LINES INTERSECT MIN. DIST=0.0
	ACUTE ANGLE = XX.X OBTUSE ANGLE = XX.X
	Printed if the two lines intersect.
	LINES PARALLEL MIN. DIST = XX.X
	Printed if the two lines are parallel.
EXAMPLES	#n#MEA DIST:MODEL ENT d1d2

IN MODEL SPACE
NORMAL DISTANCE=1.5

MODEL ENT d3d4

IN MODEL SPACE
DELTA -X =1.4 DELTA -Y=2.4
DELTA -Z=0.7
3-D DISTANCE=4.628

COMMAND	MEASURE LENGTH
DESCRIPTION	The Measure Length command allows the user to measure the length of a contour. The curves that make up the contour may be lines, arcs, conics (ellipses, hyperbolas, or para-bolas), B-splines, C-poles, strings, or nodal lines. Successive pairs of curves in the sequence, except for the first and last, either match at endpoints or should intersect one another. The sequence of curves may form an open or a closed contour.
NOTE	Strings and nodal lines should match neighbor-ing curves at endpoints. The mechanism for handling intersections does not currently support strings and nodal lines.
SYNTAX	MEA LEN (modifiers):DRAW/MODEL ENT d1 DRAW/ MODEL LOC d2d3
MODIFIERS	
STPT	Specifies that the user will digitize a point projected on the first curve of the contour, normal to the curve. The parameter value of this projected point will be used as the starting parameter for the first curve.
ENDPT	Specifies that the user will digitize a point projected on the first curve of the contour, normal to the curve. The parameter value of this projected point will be the ending parameter for the last curve of the contour.
EXAMPLES	#n#MEA LEN STPT ENDPT:MODEL ENT d1 MODEL LOC d2d3

LENGTH=21.685911

COMMAND	VERIFY ENTITY
DESCRIPTION	The Verify Entity command confirms the existence of visible entities in the current drawing by printing all parameters and associated values for the selected entities.
	The Verify Entity command is valid at drawing level, in either Model or Draw mode.
SYNTAX	VER ENT (modifiers) (CR)
	VER ENT (modifiers): DRAW/MODEL ENT d1
MODIFIERS	
ALL	Verifies all model and drawing entities. Use the form VER ENT ALL (CR).
	The default is to verify only model entities if you are in Model mode, or only drawing entities if in Draw mode.
MODEL	Verifies only model entities, even if Draw mode is in effect.
DRAW	Verifies only drawing entities, even if Model mode is in effect.
VIEW name	Prints parameters with respect to the named view.
CPLANE	Prints parameters with respect to the active construction plane.
DPLANE	Prints parameters with respect to the plane of definition.
POSITION	In Model mode, this prints the model space coordinates of the digitized location; in Draw mode, the drawing space coordinates.
COORDINATE	Prints X, Y and Z coordinates which define the end points of each segment of a B-spline.
COEFFICIENT	Prints the coefficients of the parametric equations which define the segments of a B-spline.

EXAMPLES

#n#VER ENT:MODEL LOC d1

```
PARABOLA = ENTITY TYPE
   MODEL: COORDINATE OUTPUT
      YLO= -1.0     YHI= 1.0
      VERTEX: X=3.0 Y=4.0 Z=0.0
      CONIC PLANE FOCUS=0.35
      ROT ANGLE= -34.0

   LAYER=0     NAME=
```

|	
COMMAND	COUNT ENTITIES
DESCRIPTION	The Count Entities command allows the operator to count the total number of entities in the active part. Entities can be either visible (unblanked) or not visible (blanked). This command is valid in either Model or Draw mode and must bu invoked within an active part.
SYNTAX	COU ENT (CR)
MODIFIERS	None
RESTRICTIONS	None
ERROR MESSAGES	None
NOTE	The Count Entities command counts entities in the last database, whether they are visible or invisible.
EXAMPLE	#n#COU ENT (CR)

THIS IS AN EXAMPLE
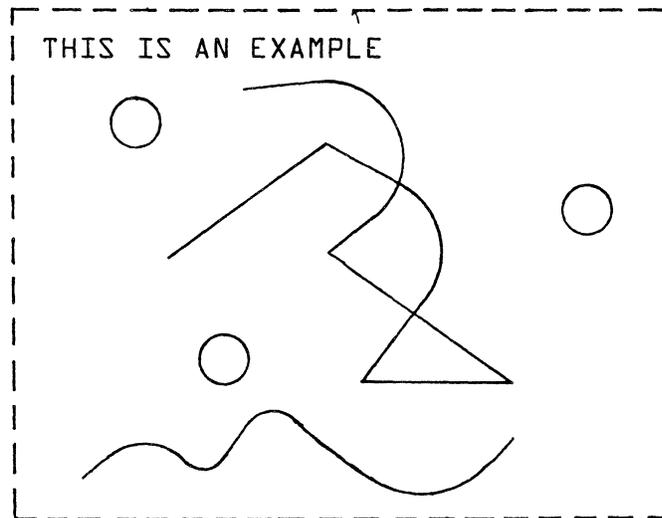
```
7 LINES
5 ARCS
1 BSPLS
1 TEXTS
1 PPE
TOTAL NUMBER OF ENTITIES =    15

PPE = PART PARAMETER ENTITY
```

NOTE	That default is the drawing origin. Drawing will be clipped using ORG as the lower left corner of the window.
WIN	Specifies window (digitized) or drawing to be plotted. Default is entire drawing.
ROT	Rotates drawing 90 degrees counter-clockwise before plotting.
DEV	Specific board to attach for plot (e.g., VSTO). Default is attached device or first board having a plotter of appropriate type.
CPY	For dot matrix plotters. Number of copies to plot. Default is 1.
STRPn	For dot matrix plotters. The number of strips to break the plot into. Default is 1.
OLPn	For dot matrix plotters. The overlap in inches between stripes. Default is 1.
SLEWn	Specifies paper advance in inches after plot is finished. Default is 9.
DENSITY'n'	Specifies shape-fill density (default is 20%).
FILE'filename'	To be used with the CV plotter only. Specifies that the plotter data will go to a file manager file named 'filename'.
TAG	Specifies that tag names, if any, are to be plotted on the CV plotter only.

COMMAND	PLOT HARD COPY
DESCRIPTION	The Plot Hard Copy command allows user to plot an image on a hard copy device exactly as it appears on an Instaview Display screen at a user-selected scale. The command is valid in all modes, but only at times when drawings are active.
SYNTAX	PLOT HARD (modifiers) (CR)
MODIFIERS	
PRIMARY	Output only primary graphics (default).
SECONDARY	Output only secondary graphics (highlighted).
ALL	Output all graphics
VERSATEC	Specifies that output is not performed on the terminal's own hardcopy device, but, rather on a VERSATEC DOT-MATRIX PLOTTER. (Default is output on the terminal's own hardcopy device.)
SCALER	Indicates that output is to be plotted at a scale of R times the scale displayed on the Instaview Display Screen, where R is a user-selected real number. (The default is Scale 1.0.)
RESTRICTIONS	1. A drawing must be active.
	2. The command can only be used at an Instaview Display Terminal station.
	3. If the VERSATEC modifier is used, a VERSATEC PLOTTER must be currently attached.
	4. This command cannot be interrupted.
ERROR MESSAGES	NO DRAWING ACTIVE, COMMAND IGNORED
	Indicates restriction 1 above was not met.
	UNIT IS NOT AN INSTAVIEW
	Indicates restriction 2 above was not met.
	NO BATCH DEVICES ATTACHED TO THE TASK
	Indicates restriction 3 above was not met.

GLOSSARY

ANGULAR DIMENSION	Used with verb "Insert" to obtain distance between angular entities. (Note: identify entities counterclockwise.)
ASSOCIATIVE DIMENSIONING	Automatic updating of dimensions as the entities to which they are linked are changed.
ATTACH MAGNETIC TAPE	Command that allows interaction between task and magnetic tape for storage and future retrievability.
BLANK	Allows for selected entities to become invisible, but not erased.
BOOT UP	Term describing restarting of system including a stop/reset, APL 1 and LOG IN.
CAD	(Computer Aided Design) Process which uses a computer to assist in the creation or modification of a design.
CADDS	Computer Aided Design and Drafting System. Trademark name of Computervision's graphic software package.
CAM	(Computer Aided Manufacturing) Process employing computer technology to manage and control the operations of a manufacturing facility.
CARTESIAN COORDINATES	The entering of explicit coordinates along the X, Y or Z axix.
CATALOG	Index which may be used to locate a particular file or group of files stored on disc or tape. More generally, a group of files belonging to one family of files.
CLIPPING	Establishing view boundaries.
CONFIGURATION	Directory of computer and peripheral devices at a single installation.
CPL	(Construction Plane) A predefined or operator-defined plane on or from which entities are located.
CONSTRUCTION SPACE	Space which uses the coordinate system defined by the construction plane.
CONTROL B	Allows operator to interrupt a command.
CONTROL L	Logs in to a task.
CONTROL Q	Aborts the command being entered.
CONTROL Rn	Same as CTRL L except that a specific task is assigned where "n" = task number.

COMMAND	PLOT CALCOMP/DOT/INTERACT/XYNETICS
DESCRIPTION	The Plot command plots the active drawing on a batch output plotter.
SYNTAX	PLOT plotter type (modifiers) (CR) Plotter types are as follows:

        DOT   Dot matrix plotter
        XYN   Xynetics plotter
        CAL   Calcomp plotter
        INT   Interact plotter

MODIFIERS	
SAME	Directs plotter to use the values specified in the previous plot command instead of default values for plot. If this modifier is received at all then the setting of default values is skipped. Any other modifiers received will set the appropriate values as usual. For example: You may enter the SAME modifier and still input SCL.
NOTE	To effectively use the SAME modifier, the device must be explicitly attached using the Attach Device command. Once the plotting is complete, the device must be explicitly detatched using the Detatch Device command. Please refer to the Attach and Detatch Device on-line documentation files for syntax and modifier usage of these commands.
SCLn	Scale to plot (default is the true drawing size).
EXT	Scales the plot to the extents of the plotter.
PEN#,lay list	(Pen Plotters only). Pen number followed by list of layers associated with that pen. If just the pen number is received then all visible layers are plotted with that pen. (Default is pen1.) This modifier may be specified more than once for different pens.
ORG	Specified position on drawing (digitized) will plot at the plotter origin if it exists or at the start pen position.

CROSSHATCHING	Process of filling in an outline with a series of symbols to highlight a part.
CURSOR	A crosshair on the CRT that corresponds to the location of the electronic pen.
DATABASE	Comprehensive collection of information having structure and organization suitable for communication, interpretation, or processing by the system.
DATA INPUT	See GETDATA.
DEFAULT	A value assumed by the system anless specifically replaced by the user.
DELETE	Command used to negate a particular construction. Commonly used with entities, group, part or view.
DIAMETER DIMENSION	Automatically measures circular entities.
DIGITIZE	The act of specifying a location or selecting an entity.
DRAWING	Drawing is a virtual piece of paper upon which the user may arrange text, title blocks, and views of the model. In CADDS 4 a part may consist of a model and many drawings. Each drawing may be of a different size or unit, and the appearance of the model in each view contained within a drawing may be independently tailored. The association between model and drawing is maintained, however, so that any changes to the model will be reflected in all drawings.
DRAWING MODE	The mode in CADDS 4 in which all entity operations affect the drawing. Drawing entities may be inserted or deleted in drawing mode. Operations performed in drawing mode do not affect the model (e.g., changes made to the model in drawing mode affect that specific drawing only- the model itself is not affected.
DYNAMIC MENU	A graphic representation of programmable commands which are defined and formated by the user and displayed on an Instaview screen. Command keywords are accessed by digitizing.
DYNAMICS	The ability to move or insert a set of graphics on the screen under control of the cursor or the ICU.
ECHO	The command that allows the user to see certain system parameters (i.e., grid, layer, clipping boundaries).
EDIT	Allows for changes, additions, or deletions in text.

END POINT	A coordinate value (X,Y,Z) representing the end of an entity.
ENTITY	Fundamental building blocks which a designer uses to represent a product, e.g. arc, circle, line, text, point, B-spline, etc.
ENTITY MASK	selection of specific points on various entities.
ESCAPE	Special key allowing user to interrupt computer operation (see Control B).
EXECUTE FILE	A file program containing a set of system and/or CADDS commands wich will automatically be executed by the system.
EXIT	To terminate work on a part.
EXTENTS	The amount of space or surface that something (i.e., part, entity, view) occupies or the distance it extends on the CRT.
FILE	A logical collection of data treated as a unit which occupies space on a storage device such as a disc.
FONTS	Kinds of type, all of one size and style. Text font: a complete character set of one style and size. Line font: a repetitive pattern used to give meaning to a line.
GETDATA	General purpose means of inputting CADDS system data. GETDATA will prompt the user by indicating the type of input that is required to obtain a desired action by the system.
GRID	Network of uniformly spaced points.
GROUP	A number of entities identified by the system as one.
HARDCOPY	A paper copy of computer output.
ICU	(Image Control Unit) The image control unit is a small box a-tached to the Instaview. The ICU contains switches and buttons which control the number of text lines on the screen, background and graphics intensity, cursor appearance, zoom, scroll, hardcopy, etc.
INCRAMENTAL COORDINATES	A new set of coordinates extends from a previously derived set of coordinates rather than from the origin.
INPUT	The data to be processed, the process of putting data into the system.
INSERT	The action of placing entities, figures, or information.

INTERACTION FACTOR	When used with coordinates allows for the repetition of entities where "N" designates haw many times.
JUSTIFICA-TION	Specification of placement of text and line fonts. Text and line fonts may be left-, center-, or right-justified relative to a specified point.
KEY MENU	Present list of written commands on digitizing tablet enabling user to digitize commands rahter than typing them.
LABEL	Allows for inserting of text with a leader line from entity to text.
LAYER	Logical concept used to discriminate (separate) group(s) of data within a given drawing. Layering enables the operator to specify derived display elements to be visible. May be thought of as a series of 255 transparencies arranged in any order yet having no depth.
LINEAR DIMENSION	Automatically measures distance and/or length of lines.
LOG IN	Process whereby user enters system.
LOG OUT	Process and command whereby user exits from system.
MEASURE	Command to determine distance between two points, a point and an entity, or two entities. May also measure area or angle.
MENU	Input device consisting of command squares on a digitizing surface. A graphic representation of programmable keys that are stored on the disc.
MIRROR	Allows the creation of a mirrored image of an entity along a vector.
MODEL	A collection of data and/or geometry representing an object or product being designed.
MODEL MODE	The mode within CADDS in which all operations affect the model.
MODEL SPACE	Coordinate system in which a 3-D database is defined.
ON LINE	Equipment or devices in a system which are directly connected to and under the control of the computer.
OPERATING SYSTEM	Software program which controls the execution and implementation of sub-programs. (Systems Level)
ORIGIN	GETDATA modifier which enables user to specify the center of an entity.
OUTPUT	The end result of a computer operation.

PARAMETER	A constant whose values determine the operation of characteristics of a program.
PART	The part is the database construct which contains all pertinent information for an individual development project and is referenced by a symbolic name. The part is composed of a model and a number of drawings. Most CADDS 4 graphical operations require activation of a part before they may be used.
PLACEMENT	Assignment of an entity to a place or location.
PRECISION	When used with dimensioning allows user to indicate number of places to the right of the decimal point.
PROGRAM	Set of machine instructions or symbolic statements combined to perform a task.
PROJECT	Allows the placement of entities along the Z axis.
PROMPT	Any message or symbol from the computer system informinf or asking the user about possible actions or operations.
PROPERTIES	User-established attributes of an entity or sub-figure within the model.
RADIUS DIMENSION	Allows for the measurement of an arc or circle.
REPAINT	Command used to refresh the CRT display.
RESTORE IMAGE	Returns a portion of a drawing saved before zooming, scrolling, etc. (See Save Image)
ROTATE	The turning of entities or groups of entities about a single axis.
RUN	Continuous execution of a program.
SAVE IMAGE	Saves the current display status. Allows operator to zoom or scroll without fear of losing current status. (See Restore Image to reactivate)
SCALE	Ratio of the current display with respect to the database.
SCROLL	Temporary movement of graphic entities relative to a fixed border.
SELECT	Specifies the entity insertion paramenters. Commonly used with grid, layer, text, view, etc.
SOFTWARE	Set of programs, procedures, rules, and associated documentation concerned with the operation of a data processing system.

SYNTAX	Structure of expressions in a language and the rules governing the structure of a language.
TAG	Allows for automatic assignment of alphanumeric names to selected entities for easier identification.
TEXT	Written information in a particular font.
TEXT FILE	File containing text. Text file may be created by the user with the EDIT command or it may be created automatically by a CADDS command such as Data Extract.
TRANSLATE	To relocate an entity or group of entities to a specified point.
TRAP SIZE	An invisible area within whose boundaries the electronic pen can identify entities.
TRIM	Allows user to alter (stretch or shorten) the existing end points of an entity.
UNBLANK	Opposite of BLANK.
VERIFY	Describes the entity selected as to its length, radius, layer, and all relevant information.
VIEW	Pictorial representation of the model positioned within the confines of a drawing.
WINDOW	Bounded area which user determines for entity selection or verification.
ZOOM	To enlarge or decrease proportionally the size of display entities.

DEVICE MANAGEMENT COMMANDS

The Device Management commands allow the user to control the output of information to specific peripheral devices, such as printers and plotters. All hardcopy support is through the device management commands.

These commands include:

    PLOT DOT
    PLOT HARD