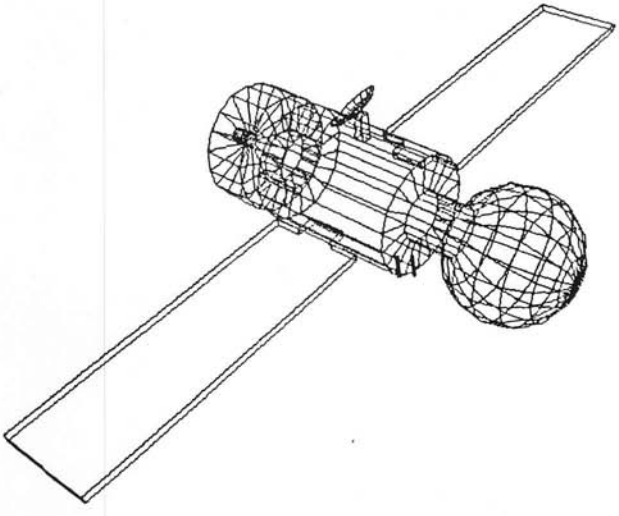
Master CAD

3-D Designer's Tool
For the Atari ST



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Master CAD

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Master CAD

3-D CAD Designers Tool

For The Atari ST Series

User Guide

Developed by I.N.D.I.
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DESIGN AND DEVELOPMENT

Main Programming and Concepts	Jaime Jerez
User Interface and Concepts	Hernan Pisani
Output Program	Luis F. Alvarez
Plotting Accessory	Rafael Santana
Languages Used	Personal Pascal™ V 1.14 TDI Modula-2/ST™

MANUAL

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Spanish Revision	Dr. Sebastian Alvarez

REVISION

Pilot Users	Guillermo Frontado Pablo Raba N.

HARDWARE REQUIREMENTS

COMPUTERS

An *Atari ST* with 1 MEG of RAM (Color or Monochrome Monitor) is required to use ***Master CAD***. A second disk drive is recommended.

PRINTERS

EPSON FX80 COMPATIBLES

You can install any G-DOS compatible driver on the ***Master CAD*** disk.

PLOTTERS

HEWLETT PACKARD™ Color Pro, 7550, 7580

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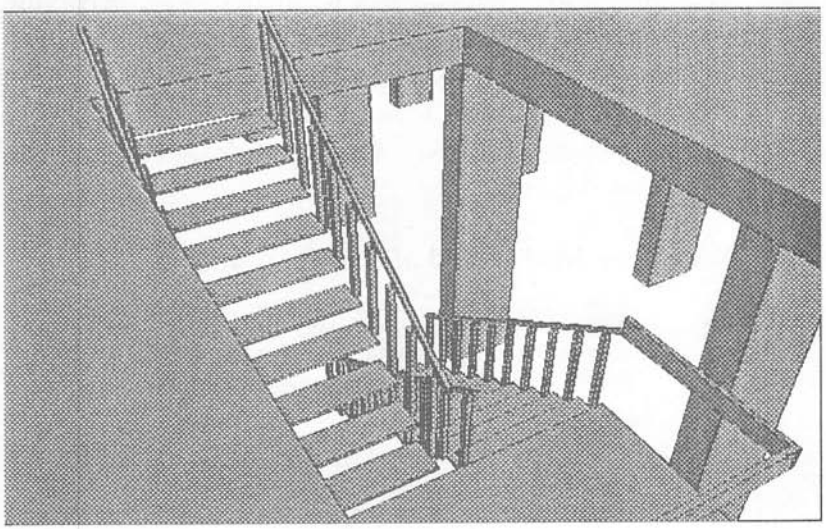
Introduction



Welcome to ***Master CAD***, *the* CAD system for the *Atari ST*. ***Master CAD*** uses a simple but revolutionary concept in graphic design which allows the user to project a design made in two dimensions into 3-D. ***Master CAD*** also offers you the ability to generate spun objects from two dimensions and project them into 3-D, revolving around the same axis. But ***Master CAD*** is more than just a design tool; it is also a system for 3-D design analysis. The program includes a complete set of "Tools", which, combined with the ease of use, makes designing a simple and enjoyable task.

With your *Atari ST* and ***Master CAD*** you can:

- Produce objects in 2-D and 3-D using pull-down menus, dialog boxes and the mouse, using very few commands from the keyboard.

- Move, copy, rotate and flip any object, horizontally or vertically; change their proportions and textures; and export or import them to and from other object-files.

- Observe the objects from various viewpoints and viewangles, external or internal, transparent or solid.

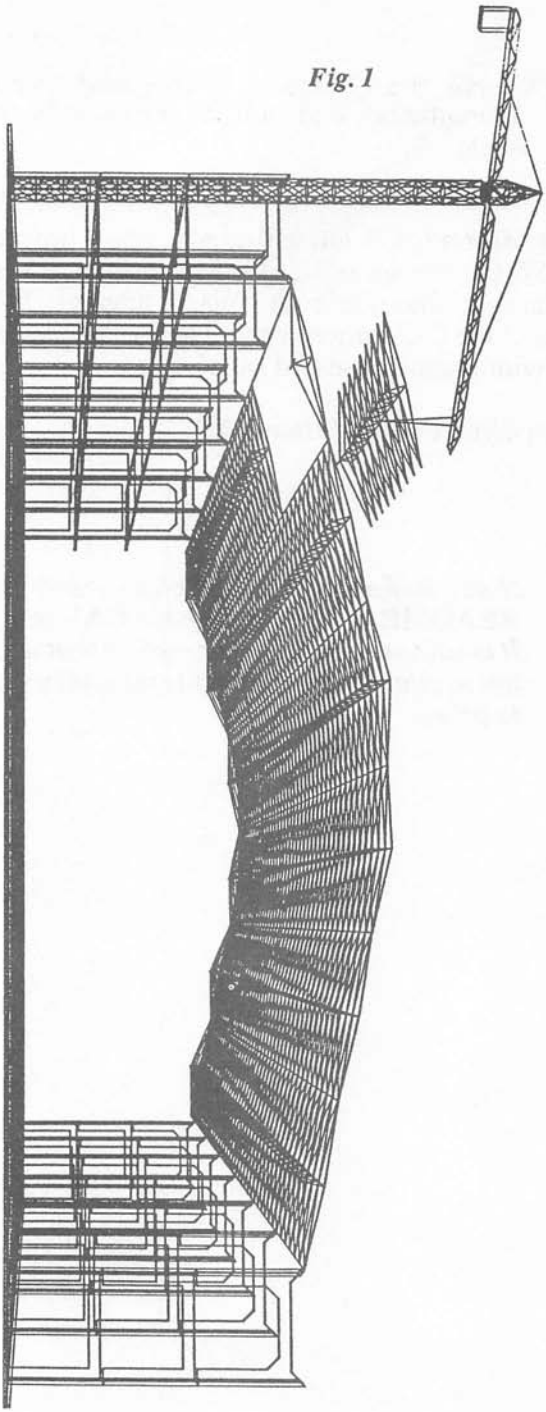
➢ Observe the objects in orthogonal, perspective, axonometric, and oblique projections from any angle.

To use ***Master CAD*** you will need to know how to use the *Atari ST*, the mouse, and the GEM interface. If you are not familiar with these, refer to your *ST* manual. Fluency in the use of the GEM interface will allow you to use ***Master CAD*** with greater ease and satisfaction.

We hope you enjoy ***Master CAD***!

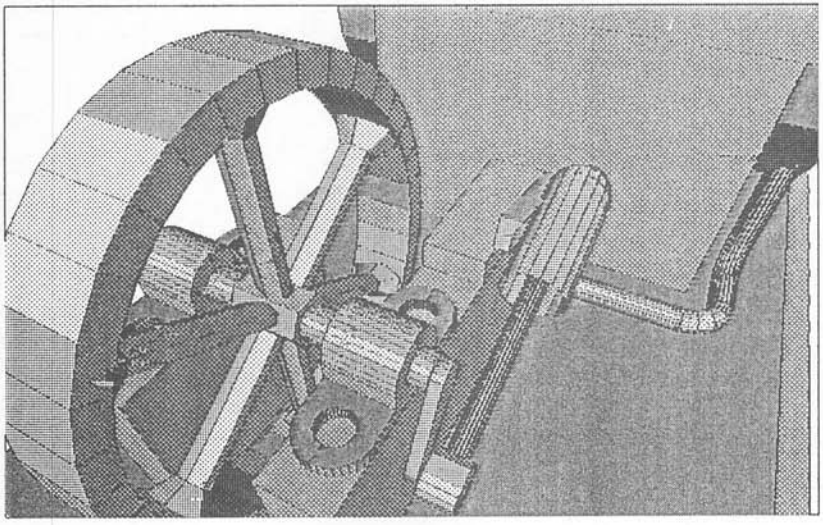
Note: Before proceeding please read the* README *file on the* Master CAD *disk. It contains updated information that was not available at the time this manual went to press.

Fig. 1



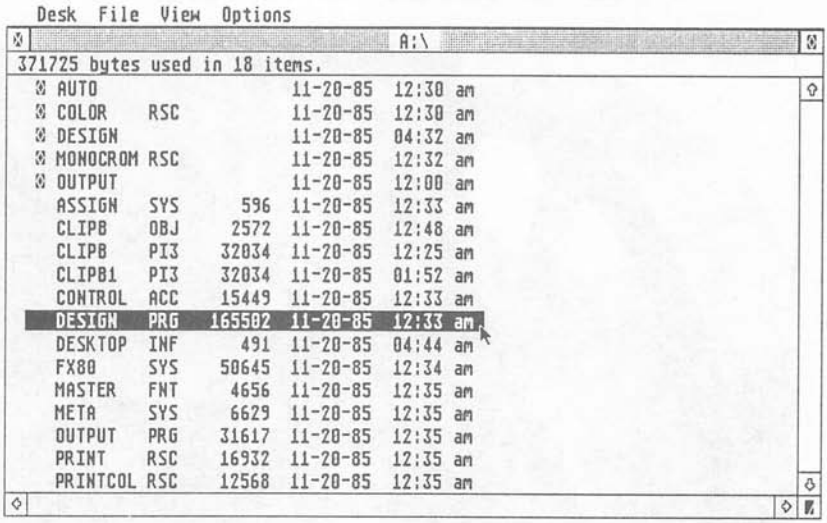
Chapter 1

Getting Started



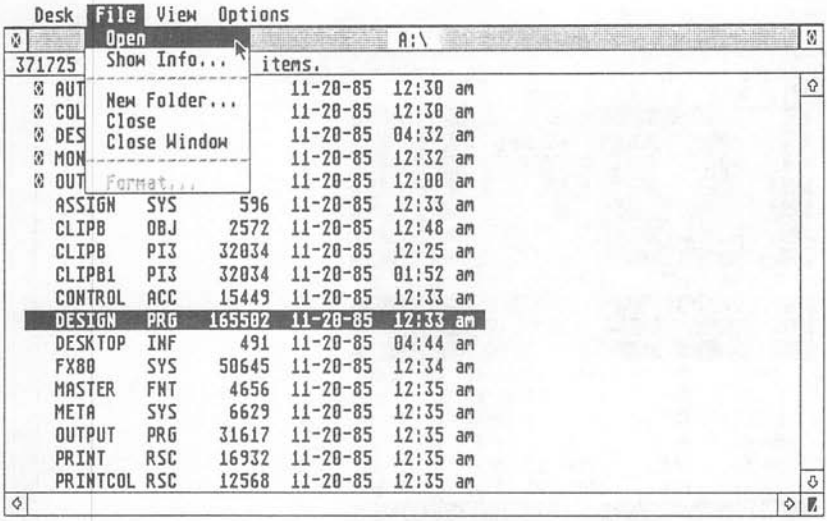
To begin, turn on your computer and wait for the GEM Desktop to appear. Start ***Master CAD***, by using the mouse to move the cursor arrow over the DESIGN.PRG icon. Click the left mouse button once. Notice that the DESIGN.PRG icon turns black to indicate that it has been selected.

Fig. 2 Click on DESIGN.PRG



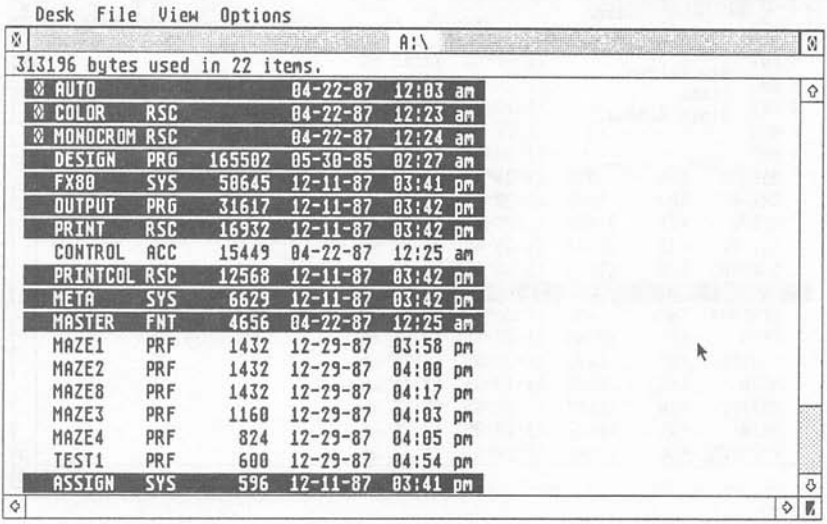
Move the cursor to the **File** drop down menu and position the arrow over the word *Open* and click the left mouse button.

Fig. 3 Select Open *in the* FILE *menu*



If you are already familiar with the *Atari ST* and the GEM Desktop, you are probably aware of a shortcut: Move the arrow cursor over the DESIGN.PRG icon and rapidly click the left button twice. ***Master CAD***'s DESIGN program will then load and run.

Fig. 4 Required files in Master CAD



The Master CAD

Drafting Board

MENU:

This is also referred to as the Menu Bar. It is found at the top of the screen, and contains all the menus available to the user at a given moment. When working on the *workscreen* board the menu bar will disappear. Run the cursor to the top of the screen and it will return.

WORKSCREEN:

The available drawing space.

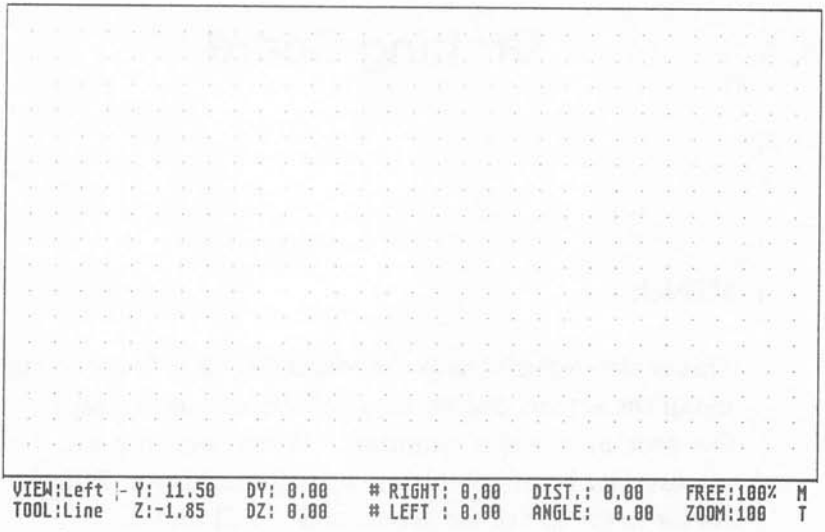
GRID:

An overlay of reference points to help in design.

MONITOR:

A display of information at the bottom of the screen that indicates where you are in the program and what is happening.

Fig. 5



The Monitor Displays:

VIEW: Indicates the orthogonal view you are working on.

TOOL: Shows the tool you are working with.

X: The absolute X coordinate measured from the origin. Also lets you know the radius of a circle or the X (horizontal) radius of an ellipse.

Y: The absolute Y coordinate measured from the origin. Also lets you know the radius of a circle or the Y (vertical) radius of an ellipse.

Z: The absolute Z coordinate measured from the origin. Also lets you know the radius of a circle or the vertical radius of an ellipse.

DX: The relative X coordinate measured from the first user-defined point. Also lets you know the radius of a circle or the X horizontal radius of an ellipse.

DY: The relative Y coordinate measured from the first user-defined point. Also lets you know the radius of a circle or Y vertical radius of an ellipse.

DZ: The relative Z coordinate measured from the first user-defined point. Also lets you know the radius of a circle or the vertical radius of an ellipse.

UP: Shows the height of the upper limit plane. If the plane is inclined, indicates the vertical variable height of the cursor position.

DOWN: Shows you the height of the bottom limit plane.

FRONT: This is the reference Y coordinate of the front limit plane.

BACK: This is the reference Y coordinate of the back limit plane.

LEFT: This is the reference X coordinate of the front limit plane.

RIGHT: This is the reference X coordinate of the front limit plane.

#: Indicates that you are working in *Cut Planes* mode.

/: Indicates that you are drawing over the *Contain Plane.*

DIST: Shows you the distance between the first user-defined point and the cursor position.

ANGLE: Shows you the angle between the line you are drawing and the horizontal.

FREE: Percentage of remaining free memory.

ZOOM: Indicates the current zoom value.

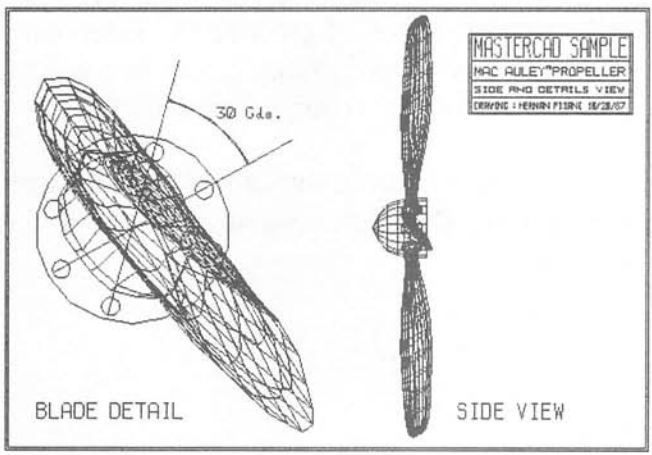
MT: Shows that you are working in Metric units of measure.

IN: Shows that you are working in English (inch) units of measure.

Now that you know more about the Drafting Board Monitor Display, let's take a closer look at it and some of the Menu Selections!

Chapter 2

Tutorial



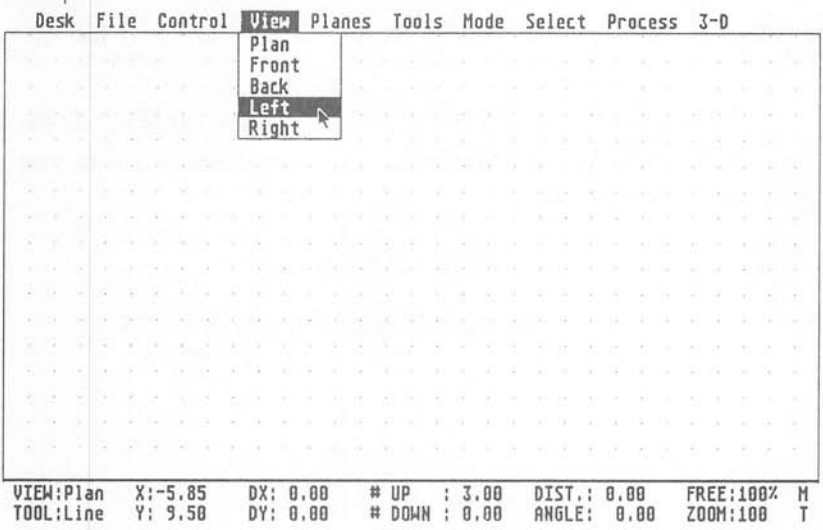
In this section of the manual, you will be introduced to the exciting world of *Computer Aided Design* (CAD). You will be given a guided tour of the different modules of ***Master CAD*** using a *structured tutorial*: we will present a series of procedures which can be used independently or as *building blocks* to form a chain of instructions for more complex procedures.

To begin your tutorial, you should be in the main work-screen of the DESIGN program (refer to page 6 of this manual).

Procedure: Opening Menus

Move the mouse and follow the cursor's movement on the workscreen. Now move the cursor to the upper part of the screen toward the Menu Bar. When the cursor touches the top of the screen the Menu Bar will appear and the nearest menu will drop down and display its contents. Position the cursor over the **View** menu. It will drop down and show the choices available to you. This procedure is quite simple and will be called *opening the* ***View*** *menu* (or whatever menu this tutorial indicates).

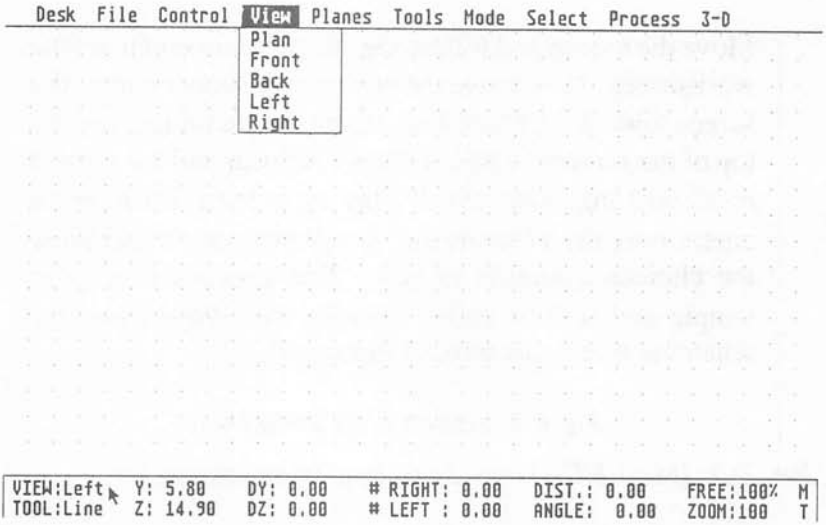
Fig. 6 Select **Left** *in the* **VIEW** *Menu*



Move the cursor over the word *Left* and click the left mouse button. For the purposes of this tutorial this action will be called just that – *click the mouse button*, or *select.*

Notice in the Monitor, at the lower part of the screen, that *Master CAD* is telling you that you are in the *Left* view mode.

Fig. 7 **Left** ***is now recorded in the Monitor as the*** **View Mode**



Master CAD has two menu bars. To access the second bar, open the **3-D** menu and watch how the menu titles change on the menu bar.

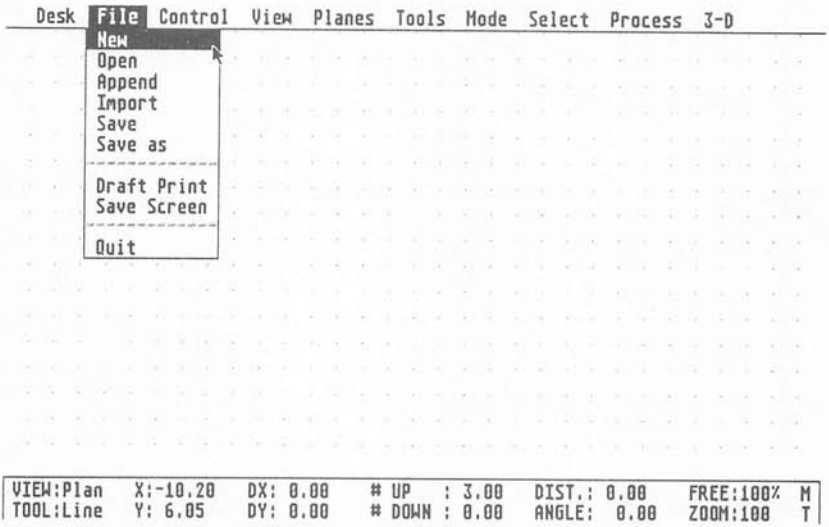
Fig. 8 2D Menu Bar

Fig. 9 3D Menu Bar

Procedure: Cube

Open the **File** menu and select the *New* option. If the *Attention* window appears, select *Continue* (Warning: This will delete anything currently in the WORK-SCREEN.

Fig. 10 Select **New** *from the* **File** *Menu*



Sub-Procedure: Normal Planes/Plan

Check the monitor window to make sure you are working in the *Plan* view. Open the **Planes** menu and select the *Normal* option. The Menu Bar will change to read *SET CUT PLANES*. Notice in the monitor that you are in the *Front* view. Move the cursor towards the bottom of the screen and watch the broken horizontal line that follows it to indicate the tracing of the plane. You now control a horizontal plane on the screen, which you may raise or lower by moving the mouse.

Fig. 11 Select Normal *From the* Planes *Menu*

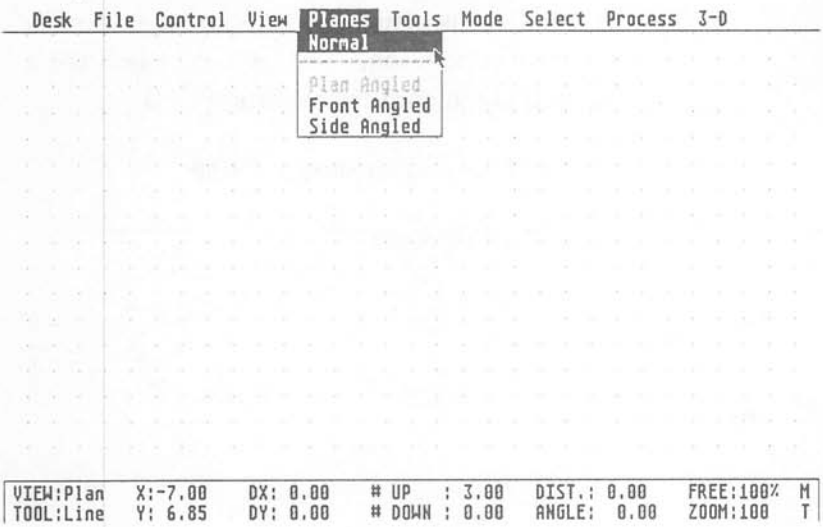
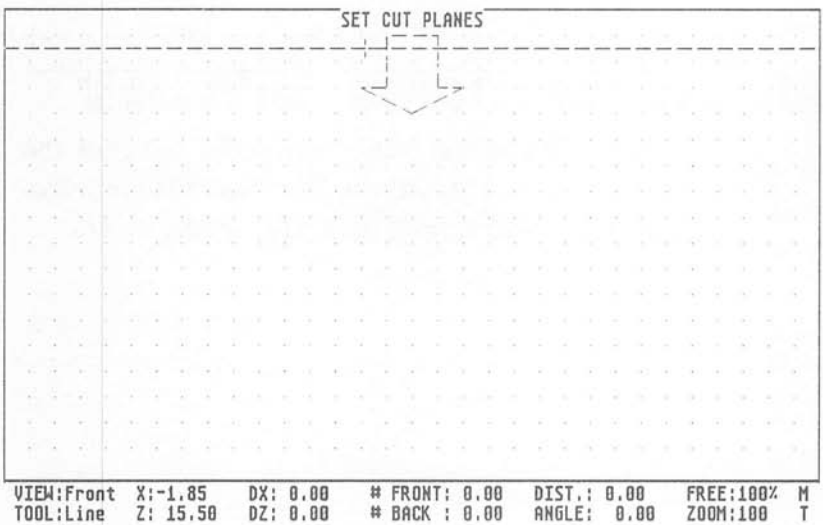
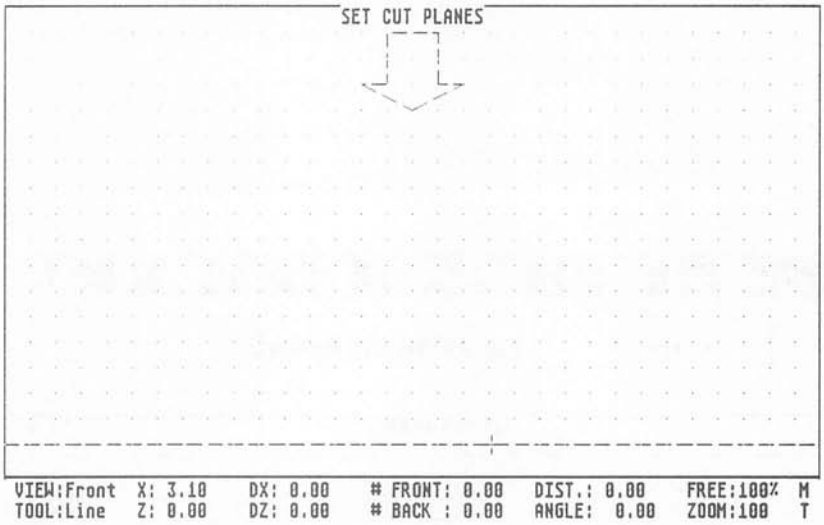


Fig. 12 Set Cut Planes...



Move the plane until the coordinate Z is equal to zero (Z:0.00) and click the left mouse button to confirm this operation. You have now defined a horizontal plane with the elevation Z=0.

Fig. 13 Set the first plane at Z:0.00



Move the second plane to (Z:4.00) and click the mouse button to confirm. *The Planes Mode Selection* box will appear. Click the mouse on *OK*.

Fig. 14 Set the second projection plane at Z:4.00

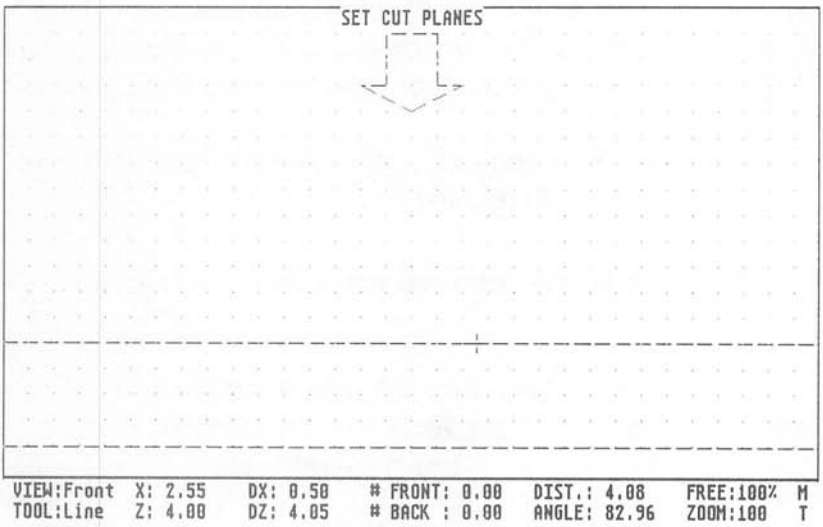
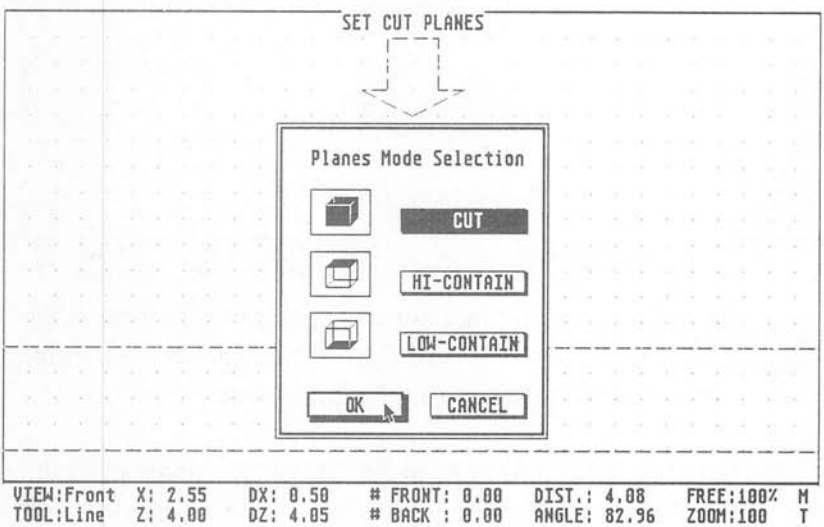


Fig. 15 Select **CUT** *from the* **Planes Mode Selection Box**

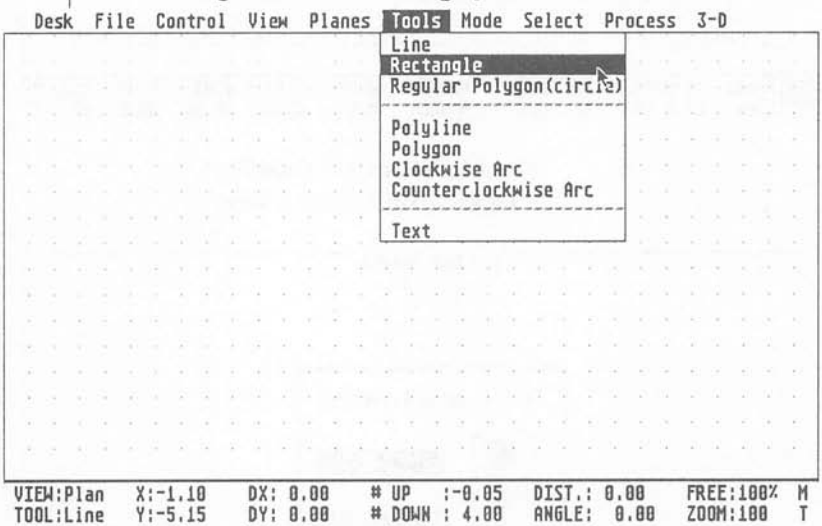


Once the planes are defined, ***Master CAD*** shows you the *Planes Mode* window. Press *OK* to use the *Cut* Mode. You now have two projection planes, with Z=0 as the base and Z=4 as the top. All drawing will be projected between these planes.

You have now completed the sub-procedure. We will continue to build the cube.

Open the **Tools** menu and select the *Rectangle* option.

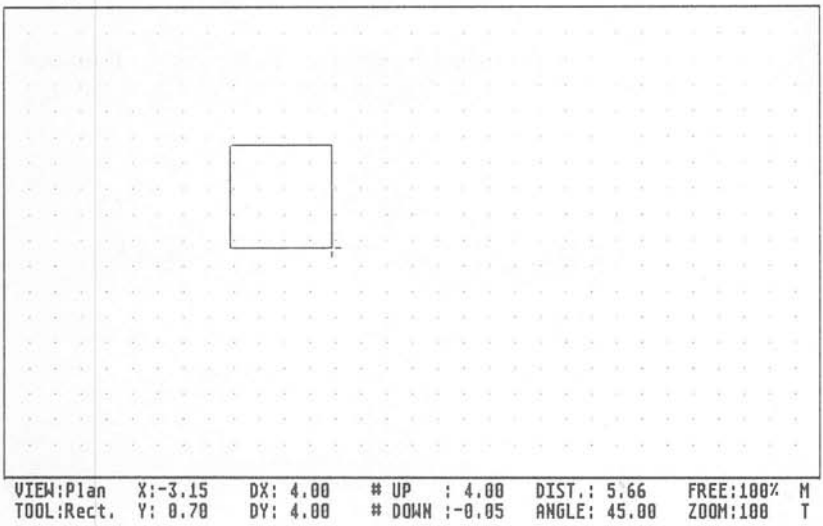
Fig. 16 Select **Rectangle** *from the* **Tools** *Menu*



Place the cursor in the center of the screen and click the mouse button to draw. Move it a short distance and notice how the rectangle is drawn on the screen.

To make a 4m x 4m rectangle, move the cursor until the monitor indicates DX:4.00 and DY:4.00, and click the mouse button to confirm. You have now defined a 4m x 4m x 4m cube, delineated from a base plane of Z=0 and with a top plane of Z=4.

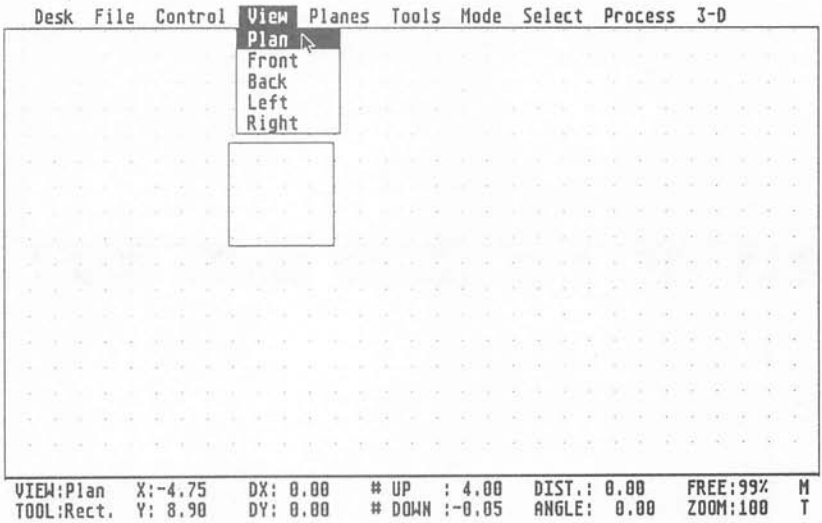
Fig. 17 Notice the **DX & DY** *reading in the* **Monitor**



Procedure: General View

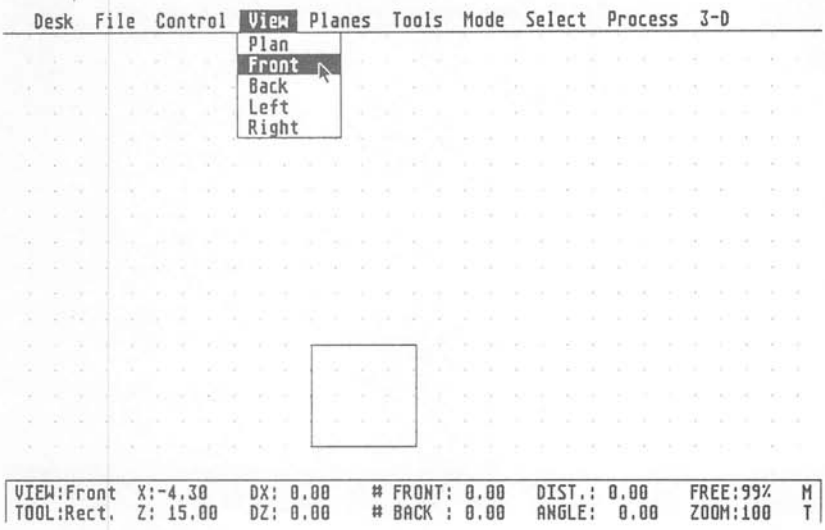
Open the **View** menu and select the *Plan* view. You will see the plan of the object developing on the screen (See Fig. 18).

Fig. 18



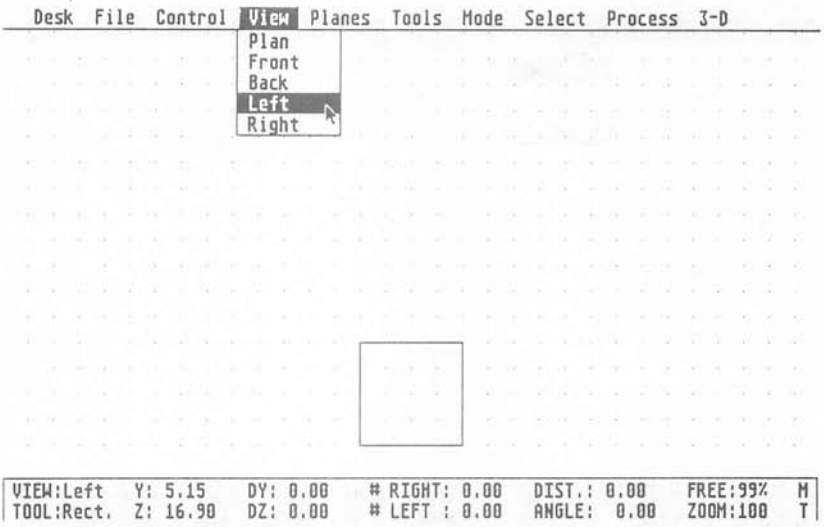
Open the **View** menu, select the *Front* view and observe (Fig. 19).

Fig. 19



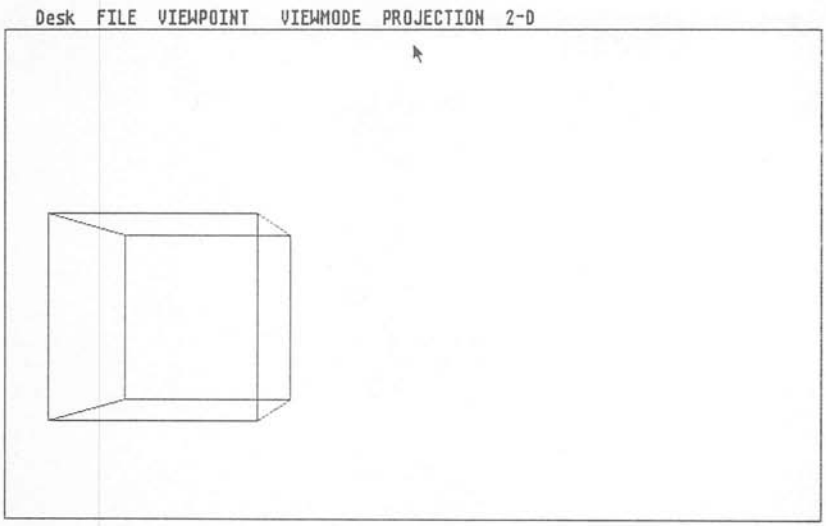
Open the **View** menu, select the *Left* view and observe (Fig. 20).

Fig. 20



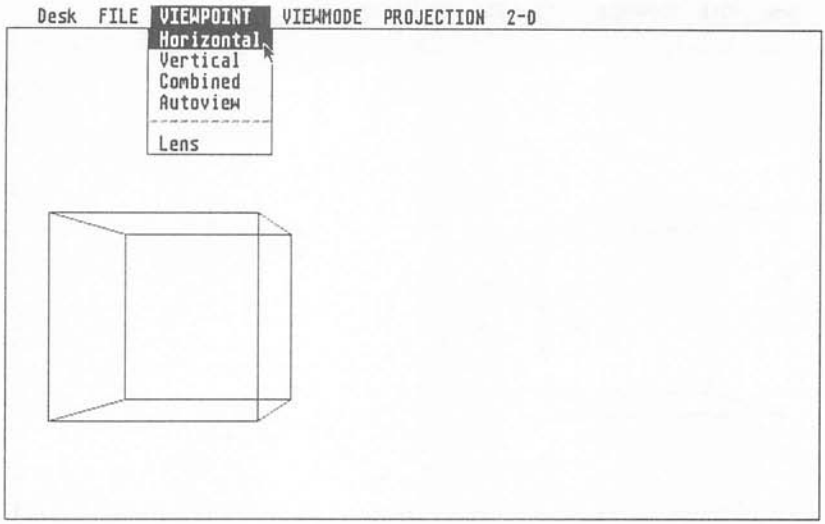
Open the **3-D** menu and observe the object in perspective (Fig. 21).

Fig. 21



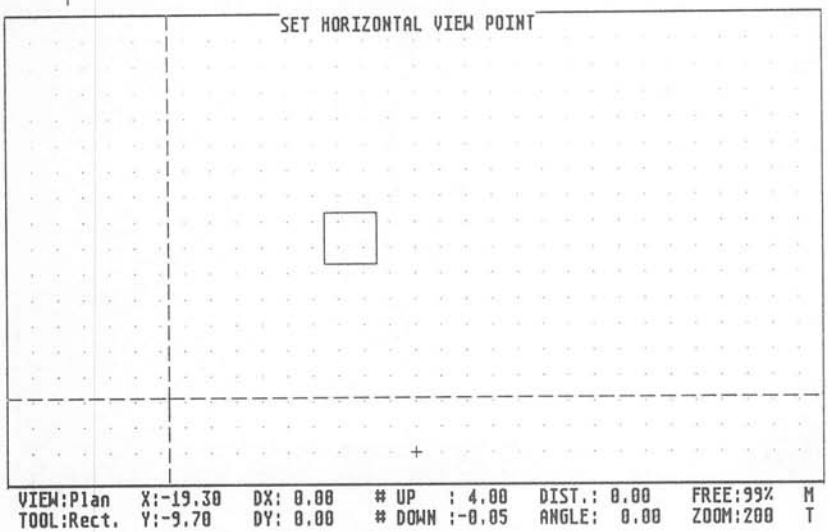
Open the **Viewpoint** menu, select *Horizontal*, and notice how the menu bar disappears and the following message appears: *SET HORIZONTAL VIEW POINT* (Fig. 22).

Fig. 22



The cursor has also been changed to a large crosshair made of intersecting broken lines. The mouse controls this crosshair.

Fig. 23 Set the Horizontal ViewPoint...



Place the crosshair center near the bottom and to the left of the object and somewhat away from it, and click the mouse button to fix the point of view. You have now set the observer's *plan* position. Move the mouse and notice how the indicator rotates according to the fixed position. Also notice the message *SET HORIZONTAL VIEW ANGLE*.

Fig. 24 ...And the Horizontal View Angle

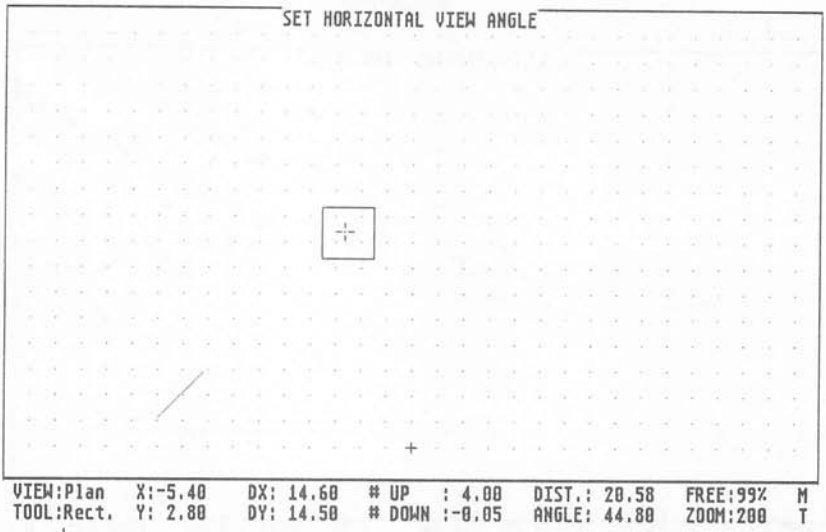
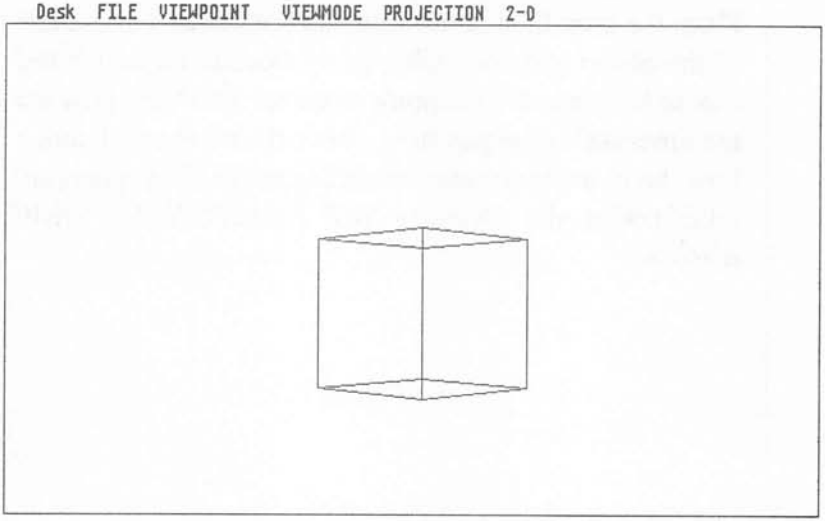
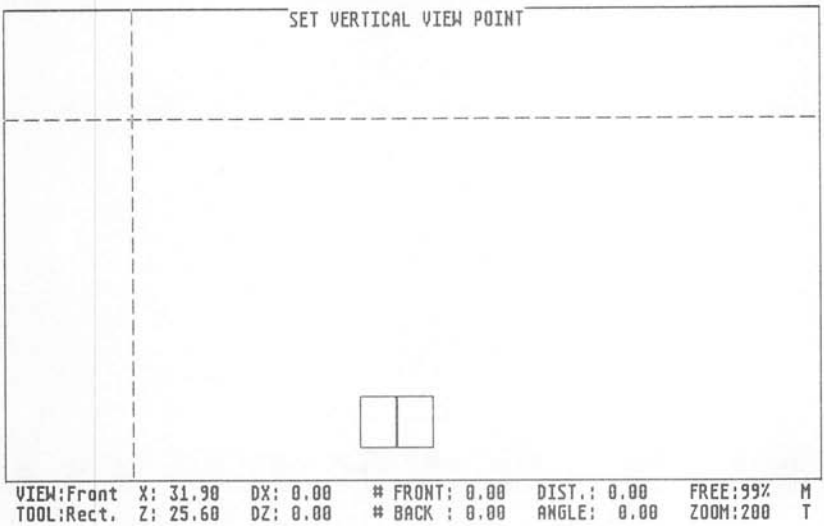


Fig. 25 And Observe the new view



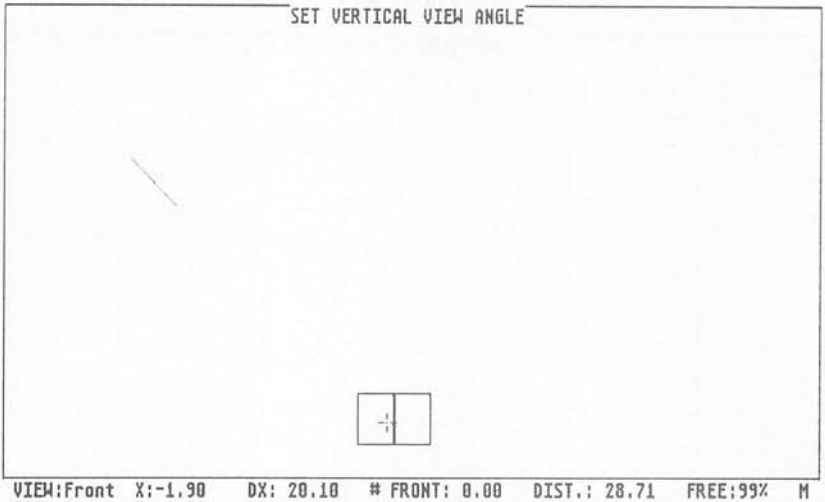
Now move the cursor inside the object to indicate the view direction, click the mouse button to set it and notice the perspective of the object developed on the workscreen.

Fig. 26 Set the Vertical View Point (the Elevation)



Open the **Viewpoint** menu again and select the *Vertical* option. An elevation of the object will appear as well as a broken horizontal line which indicates the height of the observer's viewpoint.

Fig. 27 And the Vertical View Angle



Fix the desired height, click the mouse button to set it, then aim the indicator at the object, click the button again to complete the operation, and you will be able to observe the object in perspective!

Open **Viewmode**, select *Filled Planes* and the object will change from transparent to solid on the workscreen.

Fig. 28 In the View Mode select **Filled Planes**

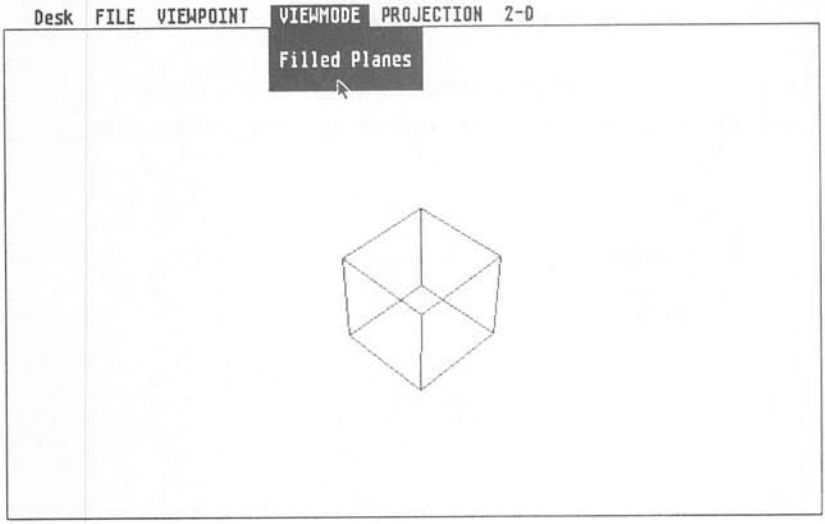
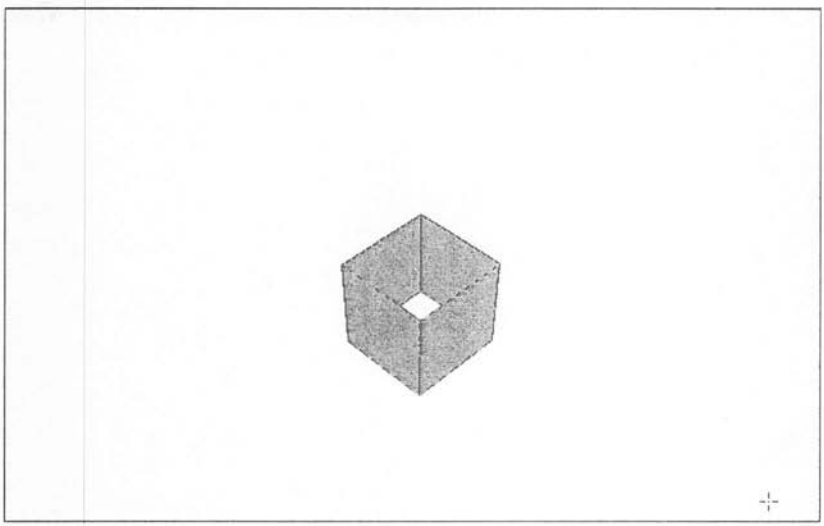
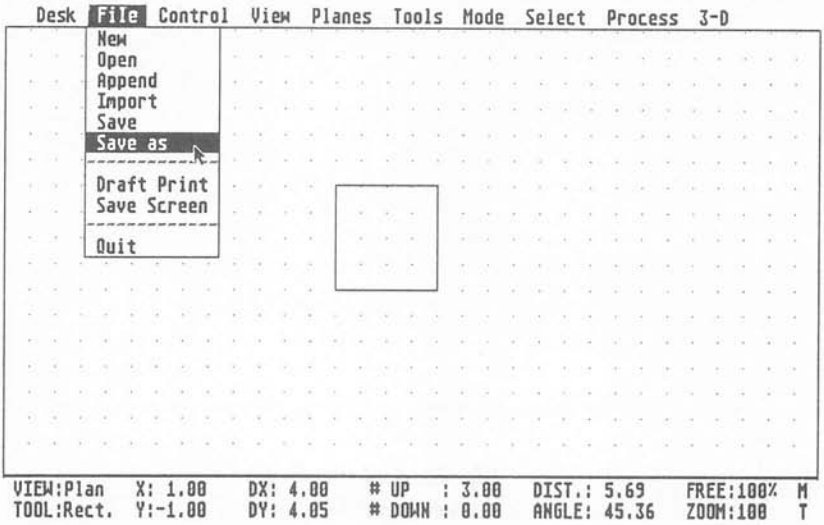


Fig. 29 And the planes take on a solid look



This ends the exercise. You may save this sample if you wish by selecting *Save As* from the **FILE** menu and naming the Object.

Fig. 30 Select **Save As** *and enter a name*



If you plan to continue after saving your work, select *New* from the **File** menu.

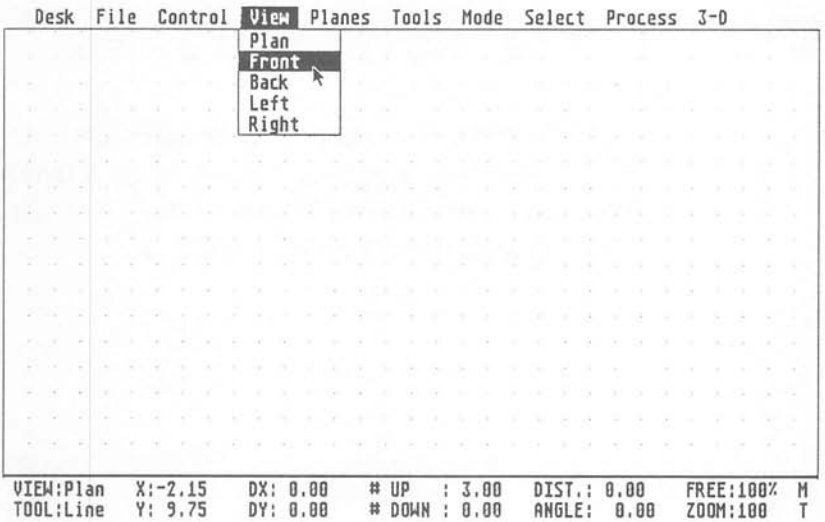
Procedure: Cylinder

Unless you are using this procedure as part of another process, open the **File** menu and select the *New* option. If the *Attention* window appears, select *Continue* (***Warning***: This will erase anything currently on the Work Screen).

Sub Procedure: Normal Planes/Front

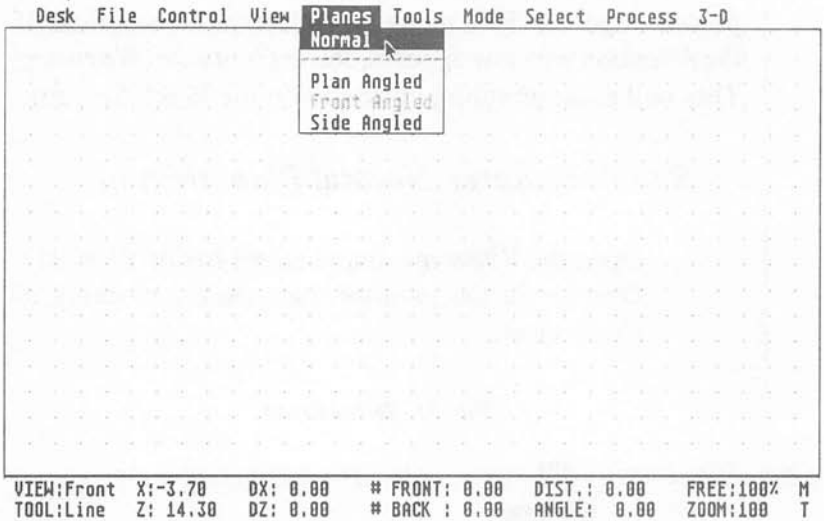
Open the **View** menu and select *Front* (Fig. 31). Observe in the monitor that you are working in *Front* view.

Fig. 31 Select Front



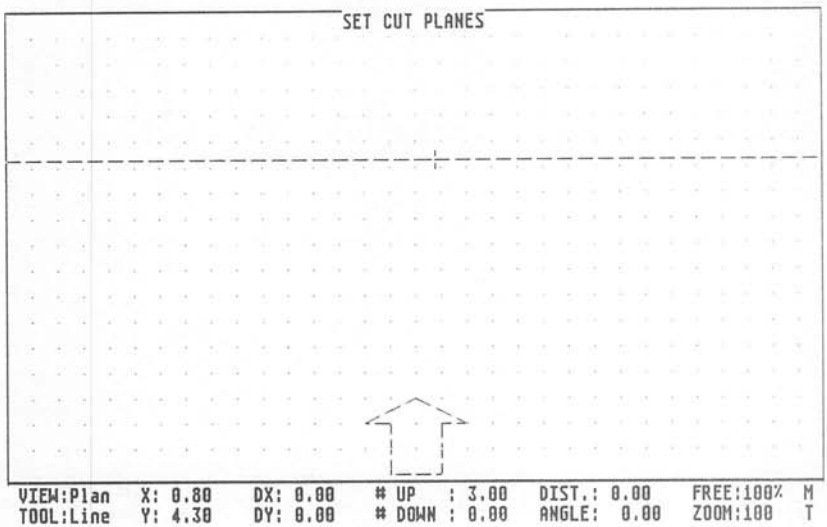
Open the **Planes** menu and select *Normal*. Notice in the monitor that you are working in the *Plan* view.

Fig. 32 Select Normal



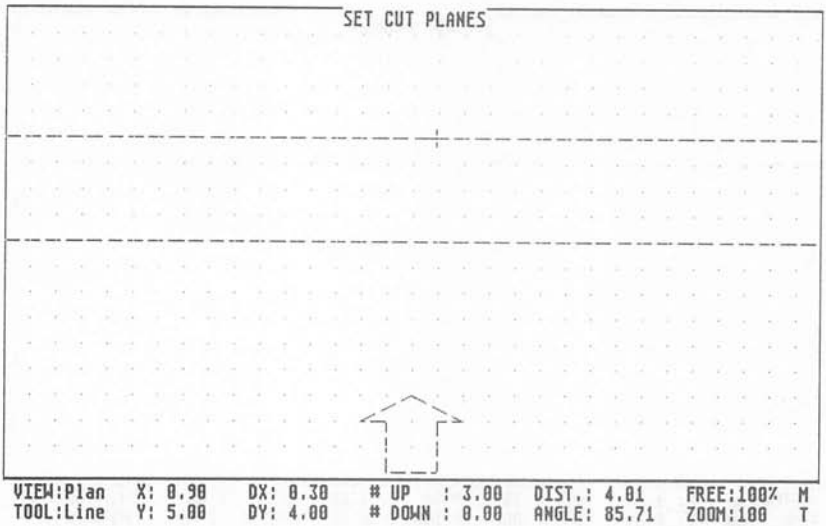
Move the cursor down and a broken horizontal line will appear, indicating the tracing of the plane. You now control a vertical plane which you can move forward or backward as you wish.

Fig. 33 Setting a Plane



Move it until the Y coordinate reads Y:1.00 and click the mouse button to set the position for this plane. You have now defined a vertical plane with the position Y=1.

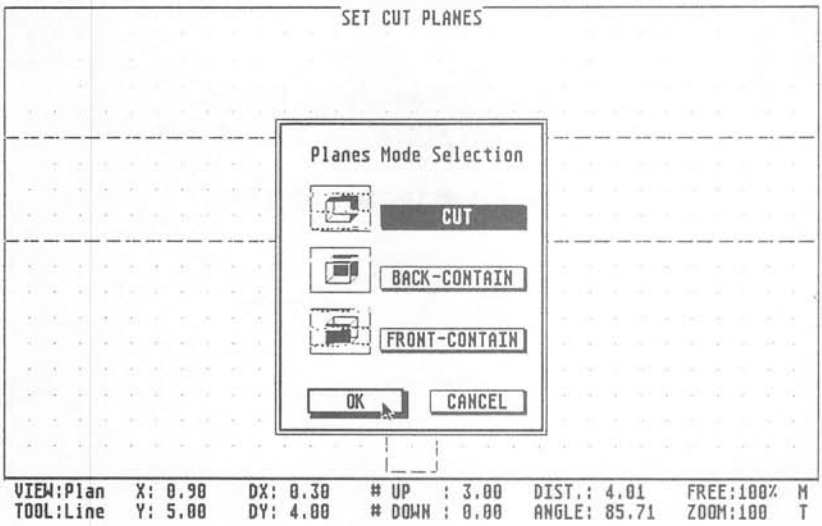
Fig. 34 Set planes at Y:1.00 and Y:5.00



Move the second plane to Y:5.00 and click the mouse button to define its position. The *Planes Mode Selection* box will appear, click the mouse on *OK*.

Once the planes are defined, ***Master CAD*** again shows you the *Planes Mode* window. Press *OK* to use the *Cut* Mode. You now have a plane at Y=5 and a second plane at Y=1.

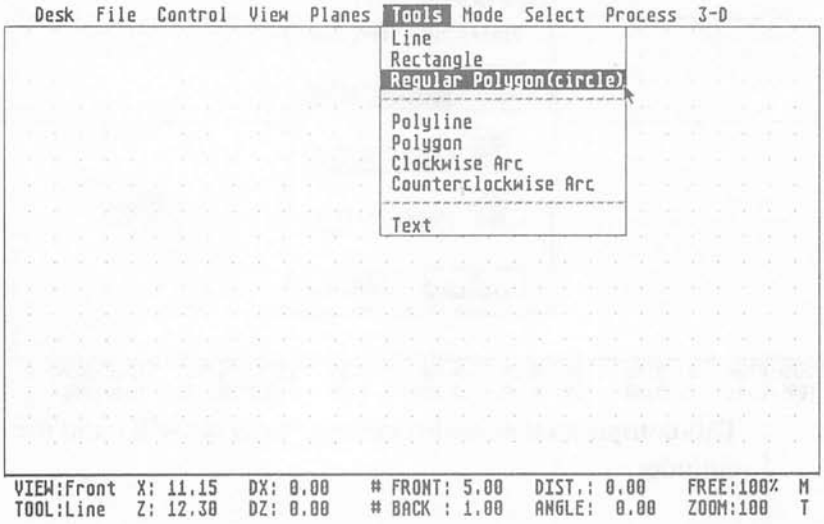
Fig. 35 Press **Ok** *in the* **Planes Mode** *window*



This completes the Sub Procedure. Now we will build the cylinder.

Open the **Tools** menu and select *Regular Polygon* (Fig. 36).

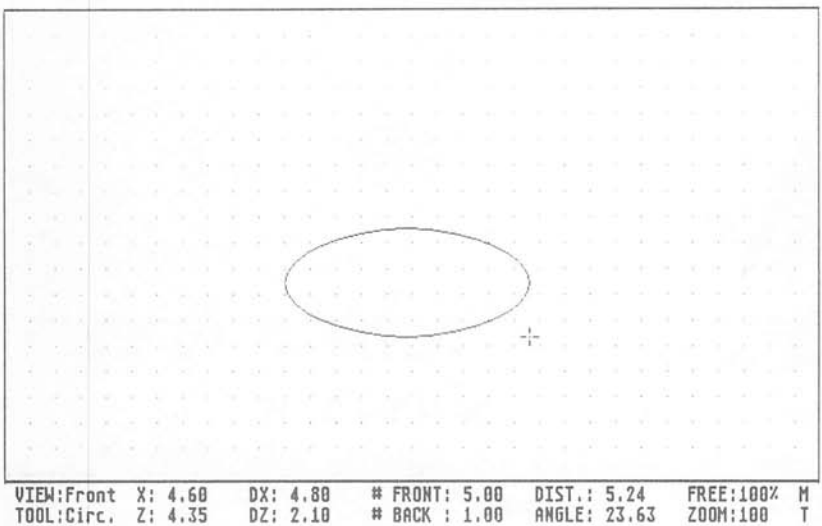
Fig.36 Select **Regular Polygon**



Place the cursor in the middle of the screen and click the mouse button once to set the center of the circle. If there is another preset object on the screen, set the circle center reasonably apart from the other figure, so that they do not collide.

Move the mouse and observe the circle forming around the center point. Choose the form and size that you wish and click the left mouse button to define the object.

Fig. 37 Drawing the Oval



Execute the procedure for **GENERAL VIEW** (refer to page 24).

Fig. 38 From **Plan View...**

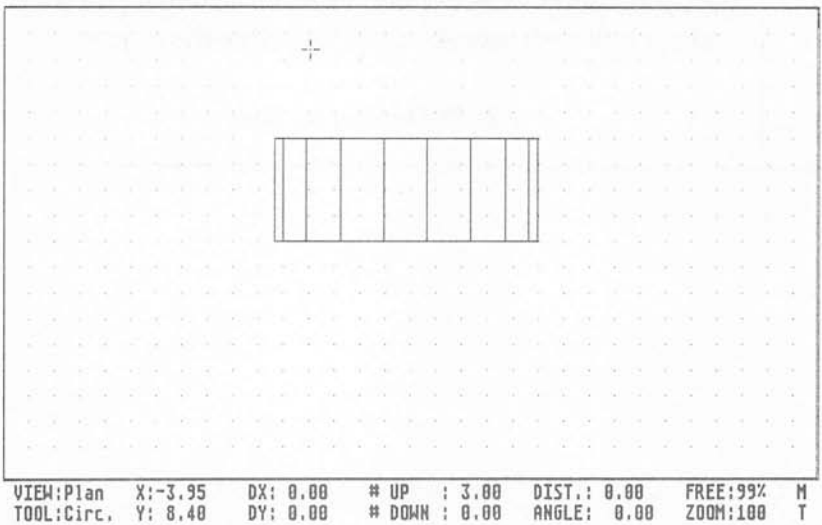


Fig. 39 Select **Front...**

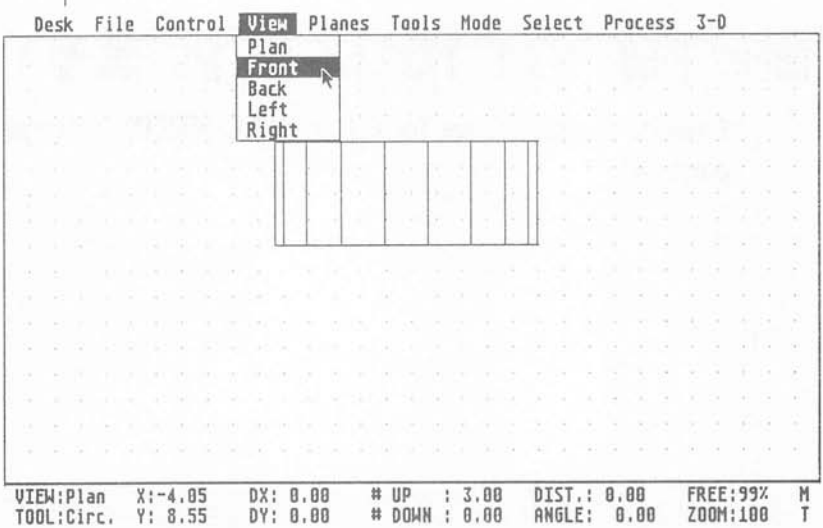


Fig. 40 ...or **Left...**

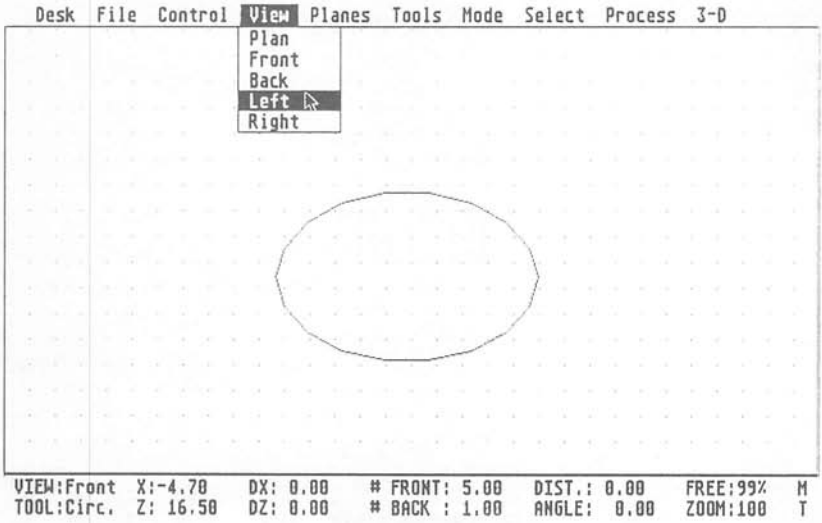


Fig. 41 To view as you wish

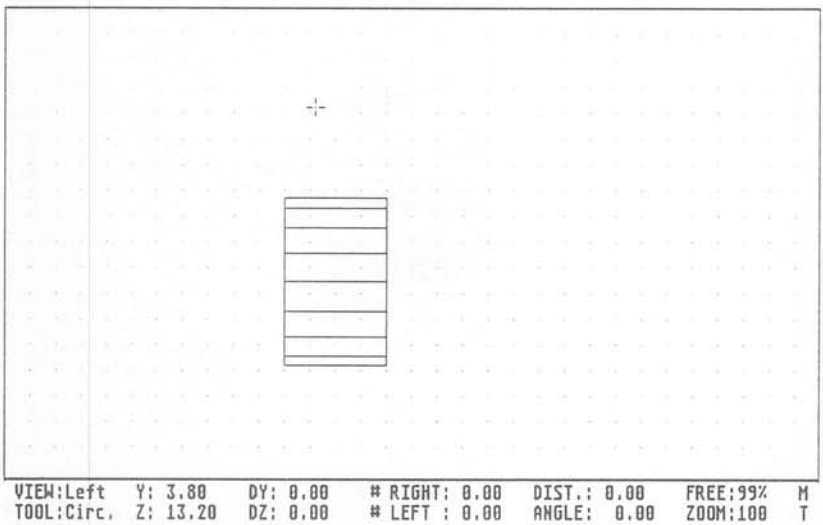


Fig. 42 Set the Horizontal View Point and Angle

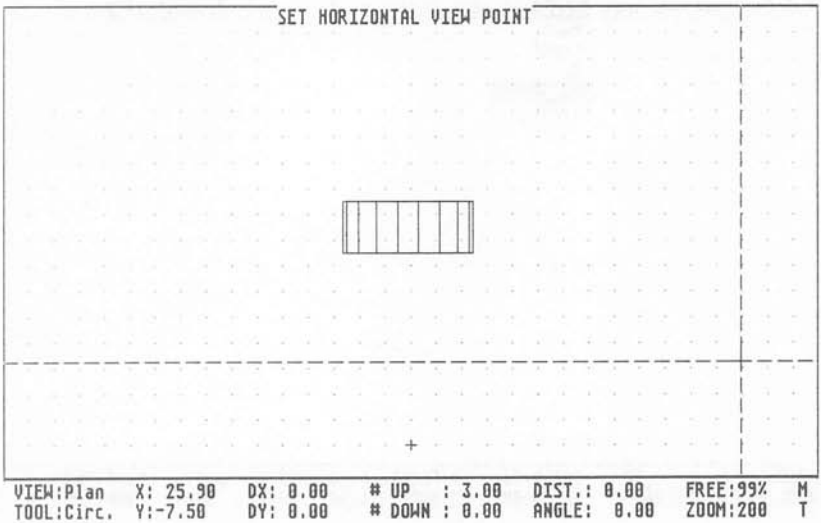


Fig. 43 Observe the results

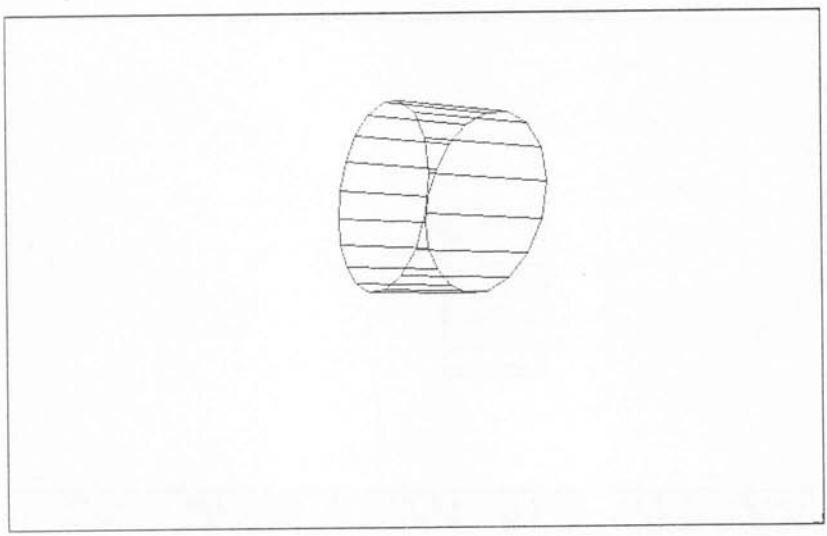


Fig. 44 Set a new Vertical View Point...

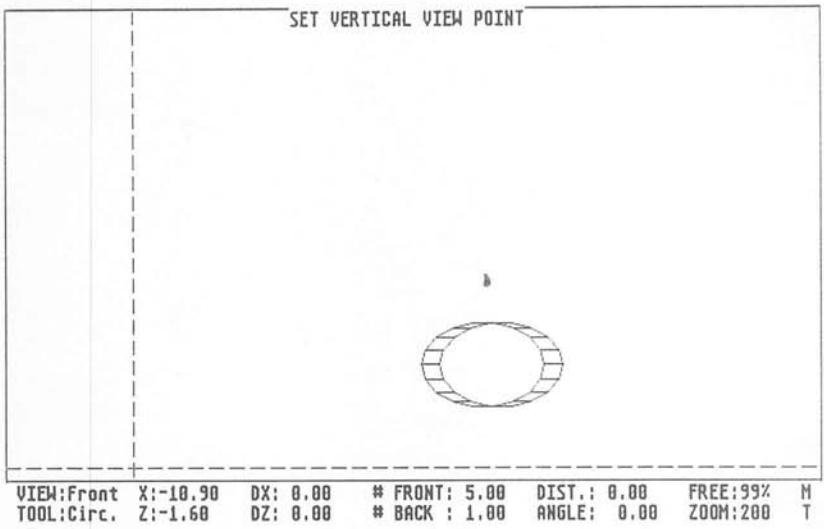


Fig. 45 And View Angle

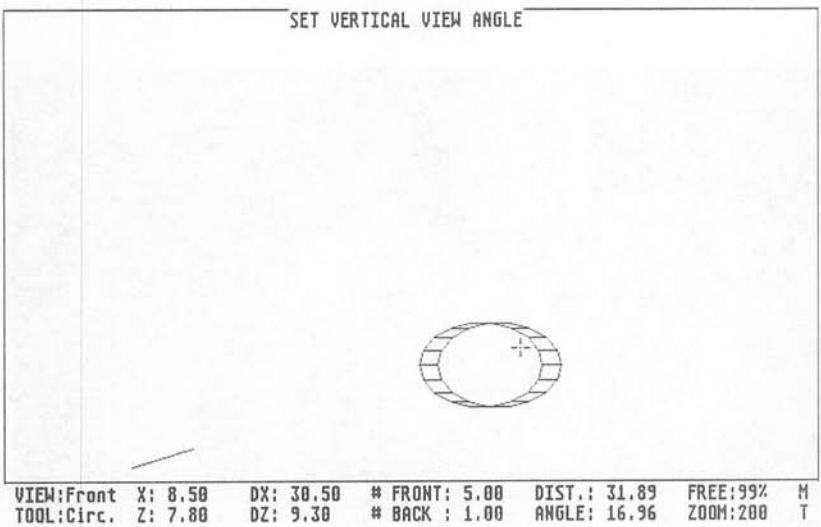


Fig. 46 ...and experiment!!

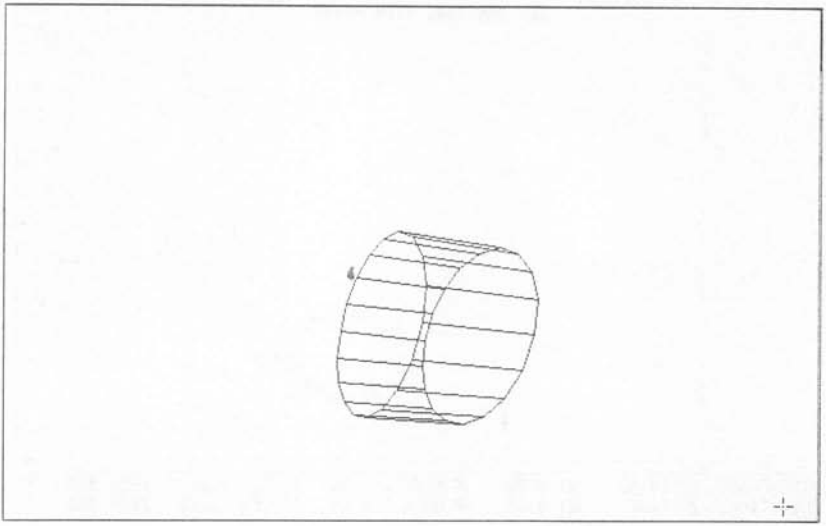


Fig. 47 Practise using different elevations...

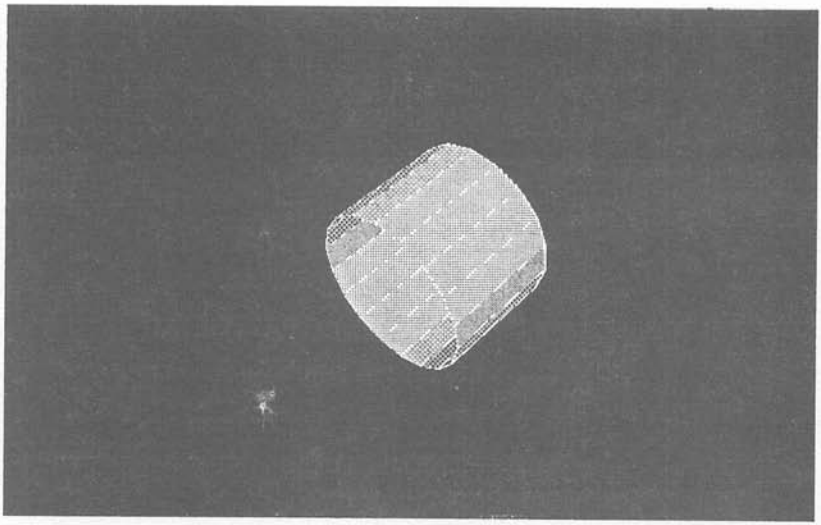


Fig. 48 ...And View Points

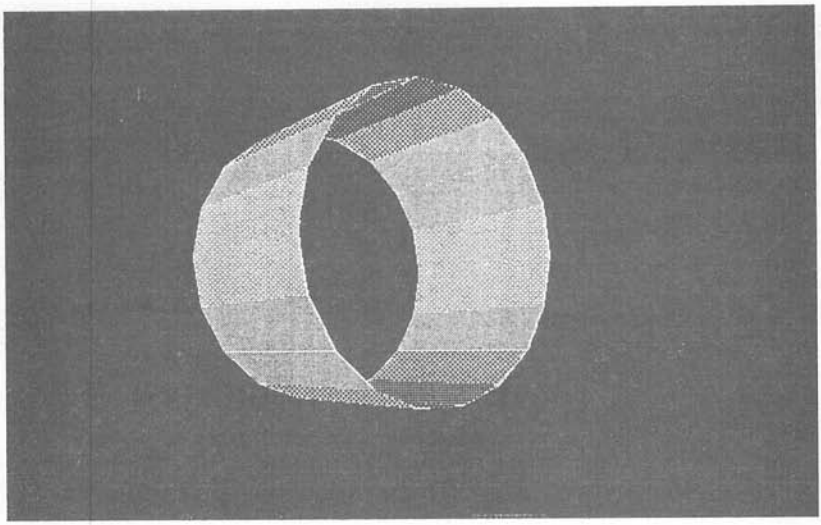


Fig. 49 ...Use Reverse Screen *from the* Control *Menu...*

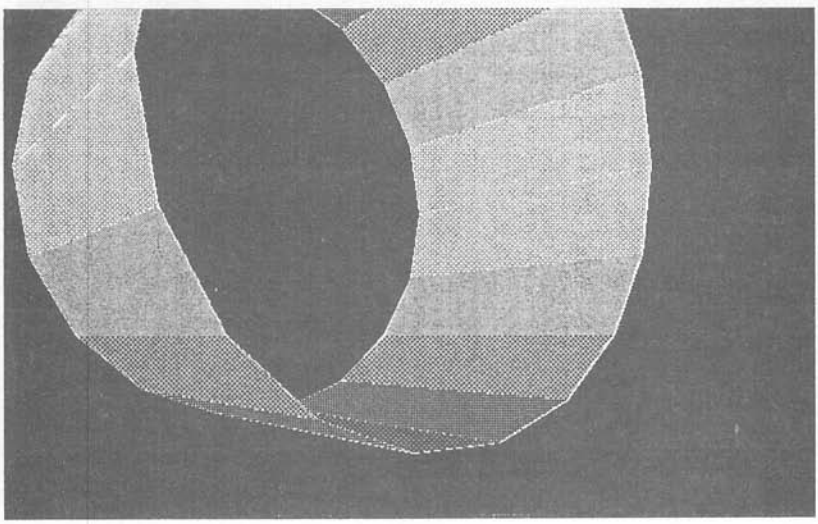


Fig. 50 ...And/or **Fill Planes** *from the* **ViewMode** *Menu...*

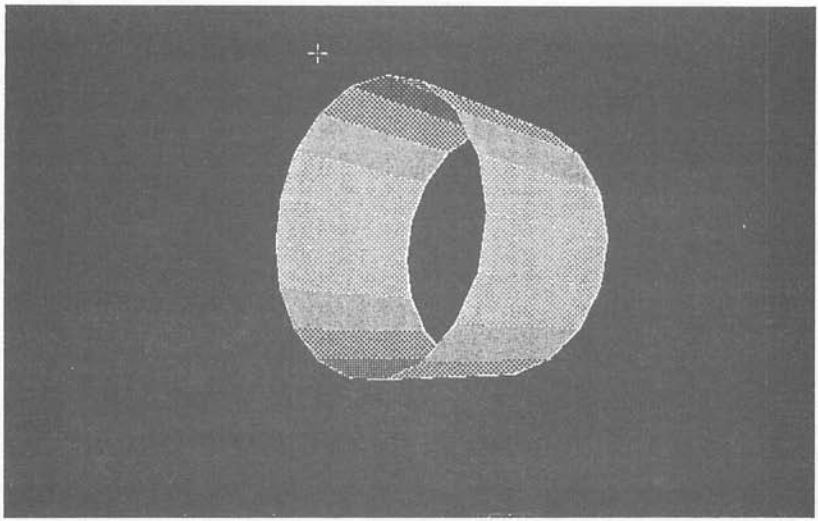
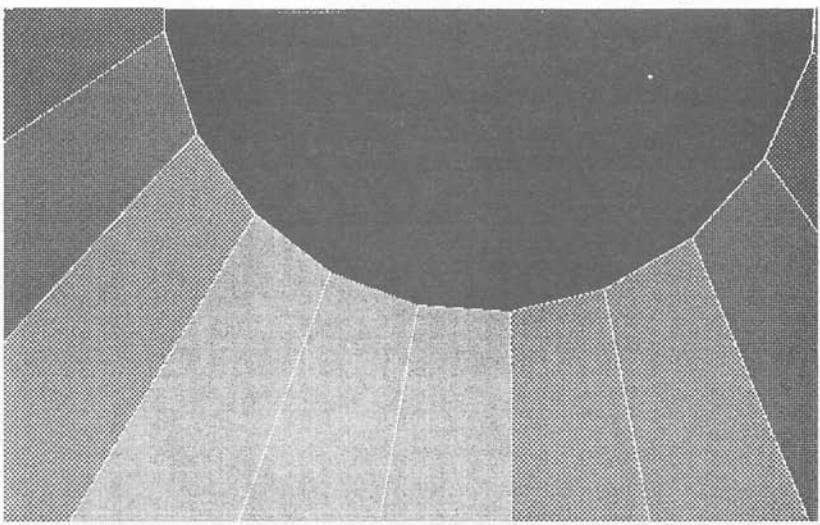


Fig. 51 ...You can even place yourself inside the cylinder!!



This ends the exercise. You may save this sample if you wish (you will be using it later in the tutorial), then continue to the next exercise. If you plan to go on after saving your work, select *New* from the **File** menu.

Procedure: Polyline

Unless you are using this procedure as part of another process, open the **File** menu and select the *New* option. If the *Attention* window appears select *Continue* (***Warning***: This will erase anything currently on the Work Screen).

Sub-Procedure: Normal Planes/Left

Open the **View** menu and select *Left* (Fig. 52). Note in the monitor that you are working in *Left* view. Open the **Planes** menu and select *Normal* (Fig. 53). Note in the monitor that you are in the *Plan* view. Move the cursor to one side and observe the broken vertical line that appears on the workscreen that indicates the tracing of the plane.

Fig. 52 Select **Left...**

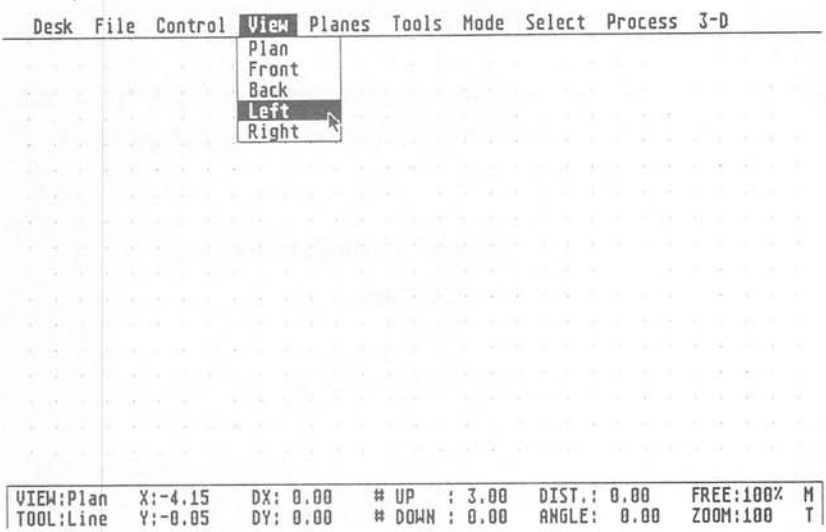
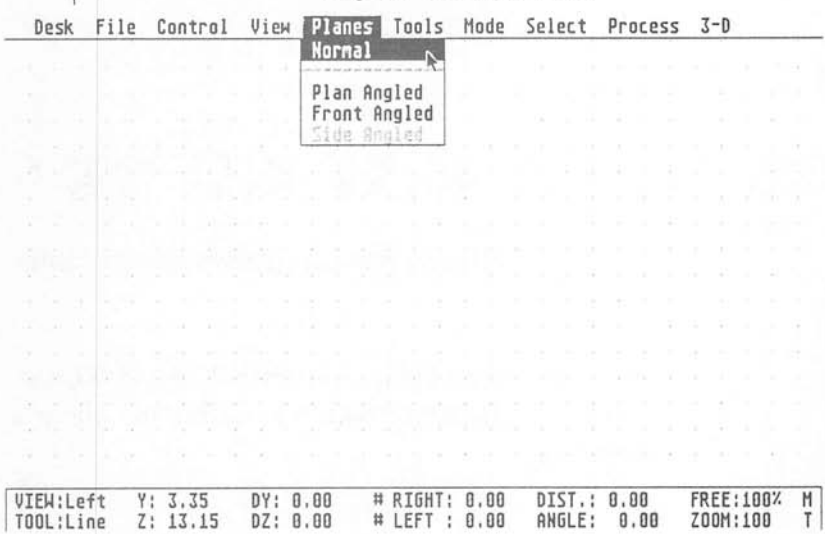


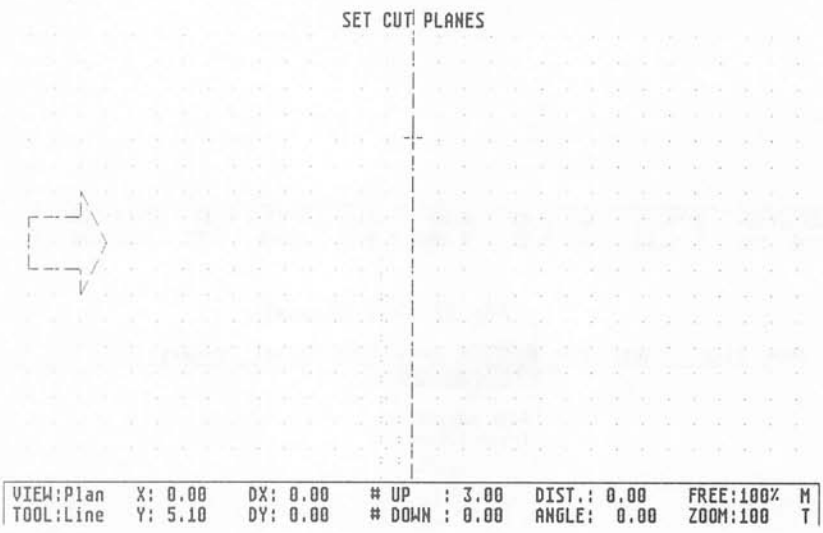
Fig. 53 Then **Normal...**



You now control a vertical plane which you may move as you like.

Move it until the X coordinate reads X:0.00 and click the left mouse button to set the position of this plane.

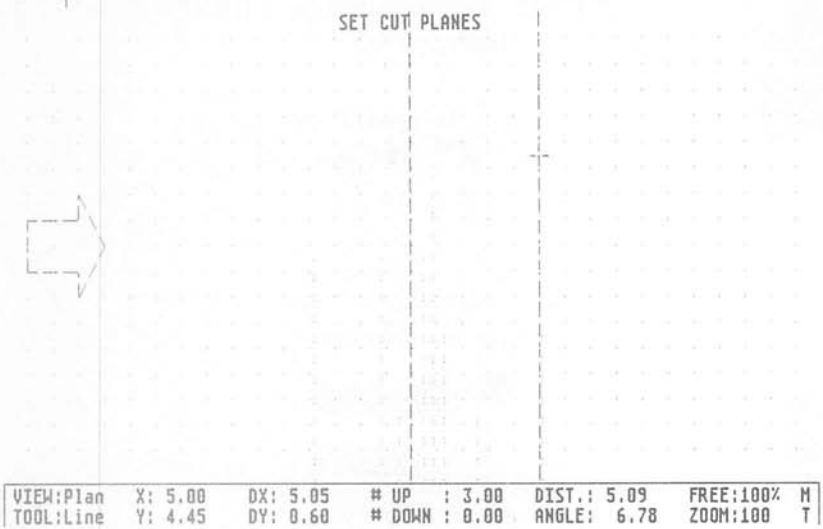
Fig. 54 Move to X:0.00 and click to set



You have defined a vertical plane with the position X=0.

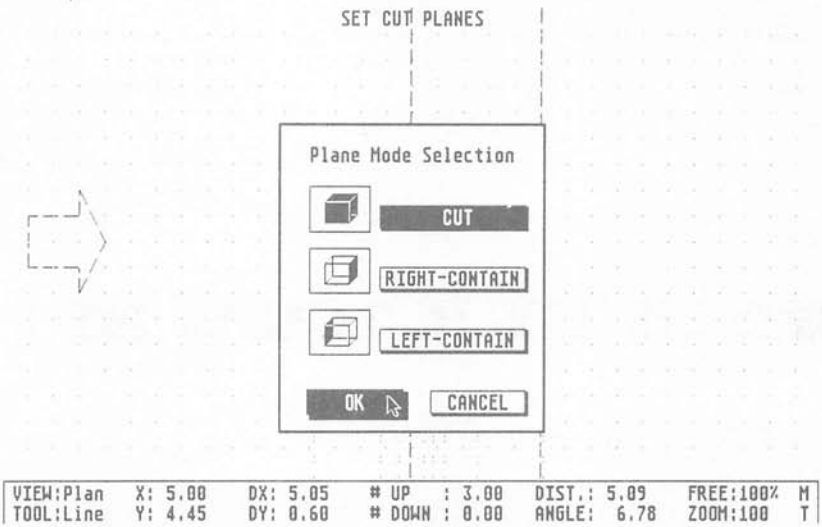
Move the second plane to the right, to X:5.00, and click the left mouse button to define the plane's position.

Fig. 55 Define the second plane at X:5.00



Once the planes are defined, ***Master CAD*** shows you the Planes Mode window. Press *OK* to use the *Cut* Mode. You have now a first plane at X=5 and a second plane at X=0.

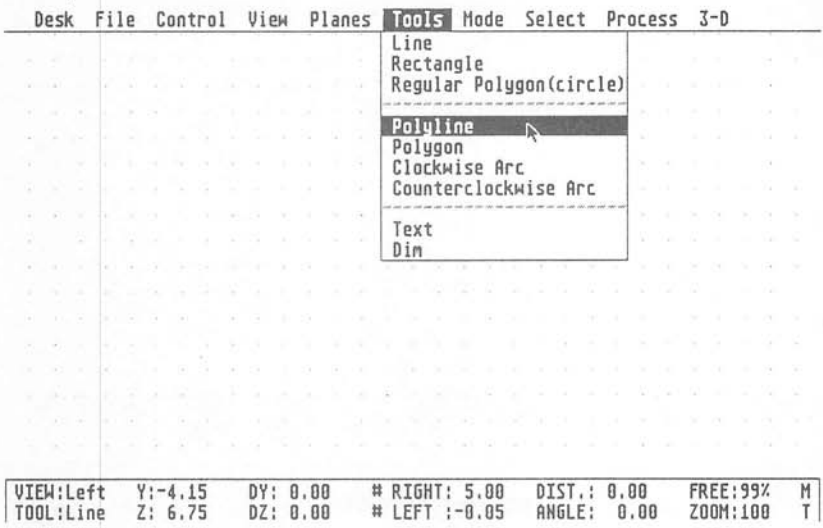
Fig. 56 Select **OK** *in the* **Cut Mode** *Dialog Box*



You have now completed the sub-procedure. We will continue to build the polyline object.

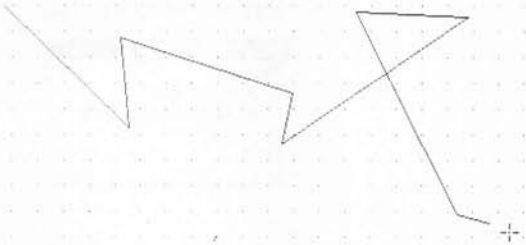
Open the **Tools** menu and select *Polyline* (Fig. 57).

Fig. 57



Place the cursor at some point on the left side of the screen and click the mouse button to begin drawing. You will notice the line forming according to its point of origin and the position of the cursor. Now move the cursor a little above and to the right of the point of origin. Click the mouse button to confirm. Move the mouse and observe the new line being drawn from the end point of the first line. Repeat the procedure as you wish. To free the cursor and complete the polyline, click the right mouse button. The line will detach from the cursor.

Fig. 58



```
VIEW:Left   Y:-10.45   DY: 0.00    # RIGHT: 5.00   DIST.: 0.00    FREE:99%   M
TOOL:Pline  Z: 2.20    DZ: 0.00    # LEFT :-0.05   ANGLE:  0.00   ZOOM:100   T
```

Execute the **GENERAL VIEW** procedure (refer to page 24).

This ends the exercise. You may save this sample if you wish, then continue on to the next exercise. If you plan to continue after saving your work, select *New* from the **File** menu.

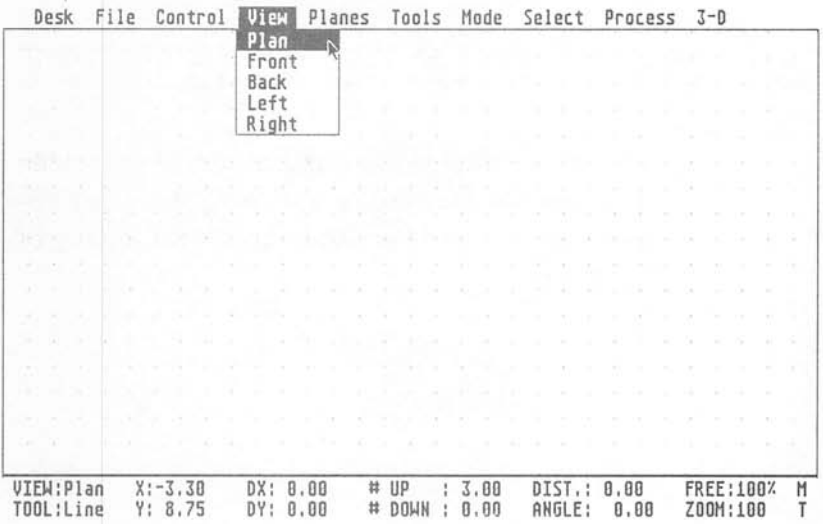
Procedure : Hexagonal Prism

Unless you are using this procedure as part of another process, open the **File** menu and select the *New* option. If the *Attention* window appears, select *Continue* (***Warning***: This will erase anything currently on the Work Screen).

Sub-Procedure: Front-Angled Planes/Plan

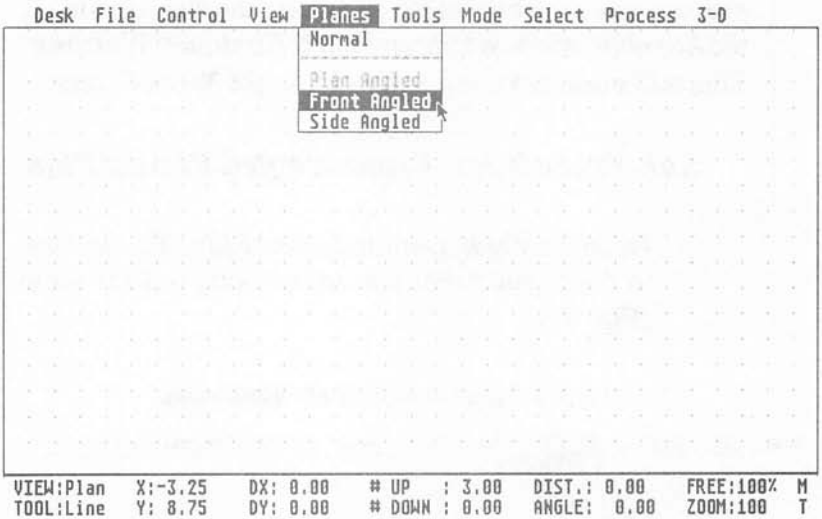
Open the **View** menu and select *Plan*. You will see in the monitor that you are working in *Plan* view (Fig. 59).

Fig. 59 Select **Plan** *from the* **View** *menu*



Open the **Planes** menu and select the *Front Angled* option (Fig. 60). Observe in the monitor that you are working in *Front* view.

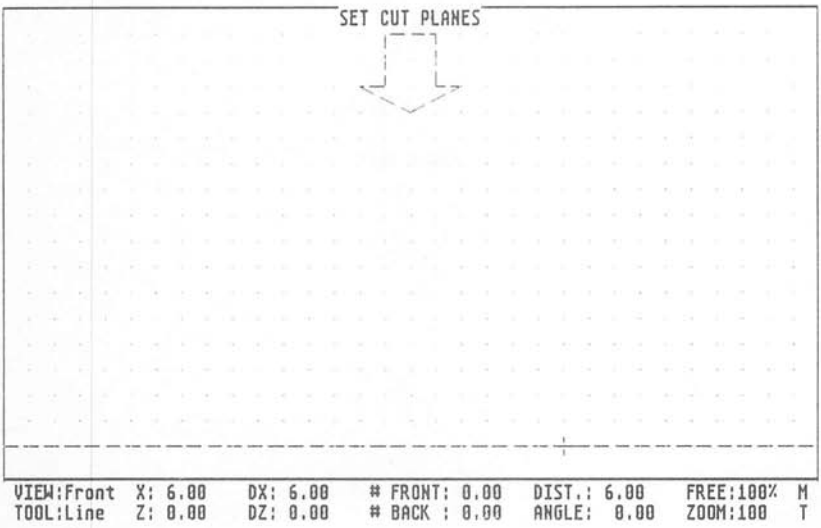
Fig. 60



Move the cursor towards the bottom of the screen and observe the broken horizontal line that appears on the screen which indicates the tracing of the plane.

You now control a horizontal plane that you can raise or lower as you wish. Move it until the Z coordinate reads Z:0.00 and click the mouse button to set the first position point of this plane.

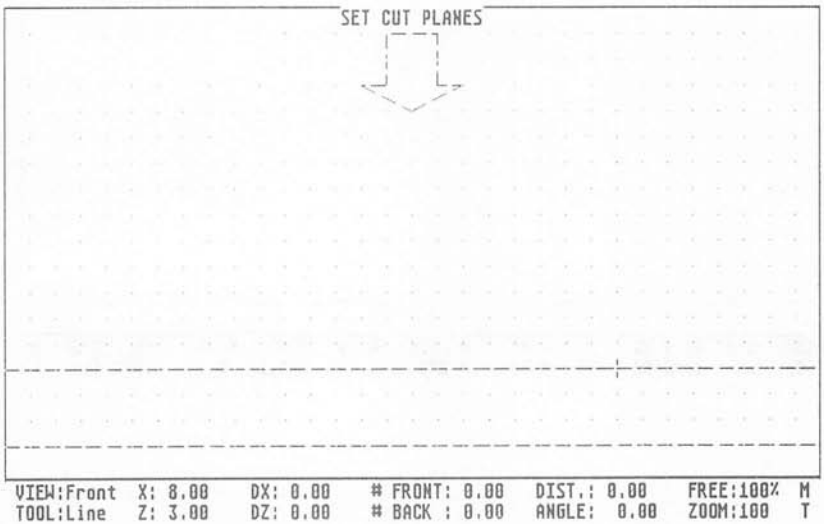
Fig. 61 Set Cut Plane at X:6.00, Z:0.00



Move the mouse and notice that the broken line rotates around the fixed point. This means that you can define the angle of this plane relative to the horizontal.

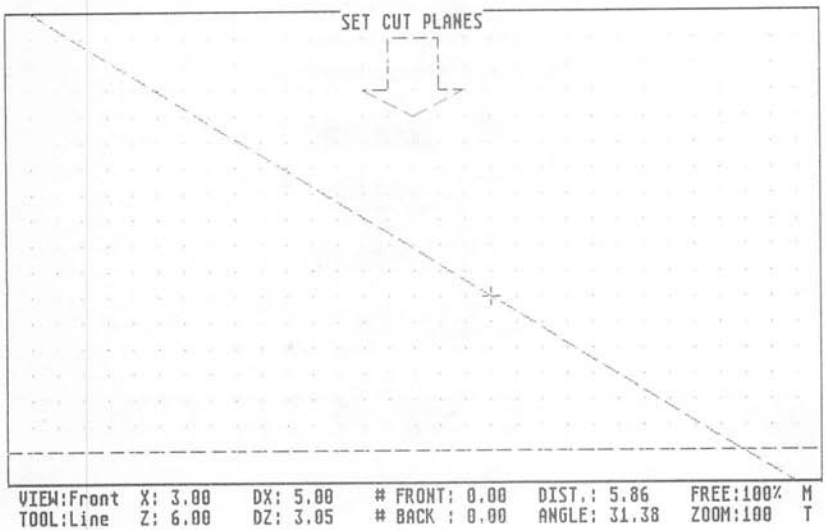
Define a horizontal plane using the cursor and set it by clicking the mouse button, set X:6.00, Z:0.00 (Fig. 62).

Fig. 62



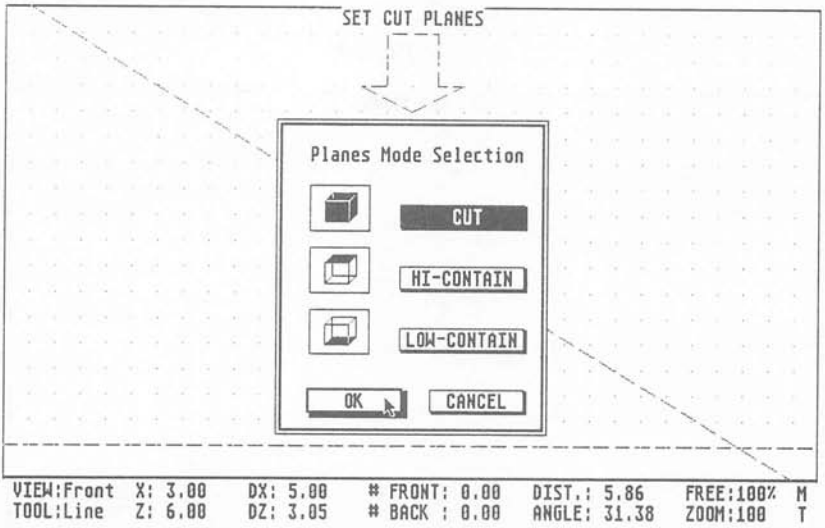
Set the cursor in position X:8.00/Z:3.00 and click the mouse button to set the first point of the second plane; then move it to X:3.00/Z:6.00 and click again to complete the operation. You have defined a horizontal plane that passes through the assigned coordinates.

Fig. 63



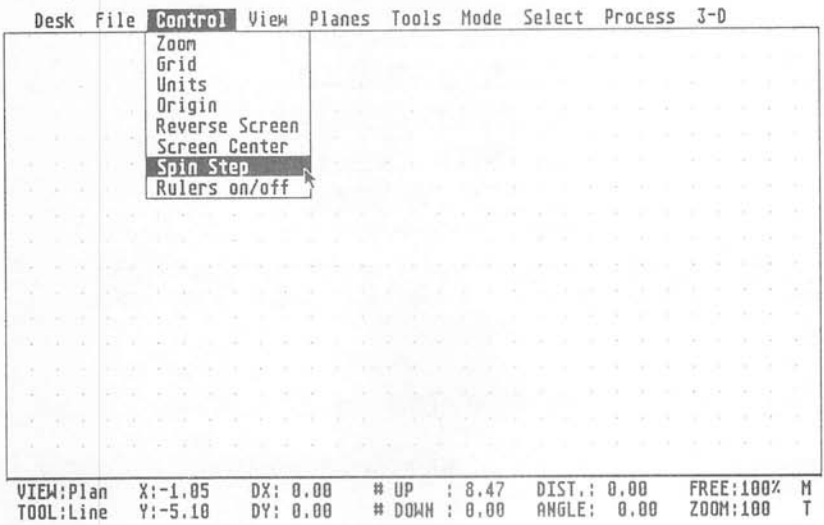
Once the planes are defined, ***Master CAD*** shows you the *Planes Mode* window. Press *OK* to use the *Cut* Mode (Fig. 64). You now have a first plane at Z=0 and a second plane of variable height.

Fig. 64



Open the **Control** menu and select *SPIN STEP* (Fig. 65). Observe the dialog box (Fig. 66) that permits you to choose a regular polygon, from a triangle to a 99-sided polygon.

Fig. 65



Choose the *Hexagon* option and press *OK* to confirm. Open the **Tools** menu and select *Regular Polygon* (Fig. 67). Place the cursor in the middle of the screen and click the mouse button to fix the center of the polygon.

Fig. 66

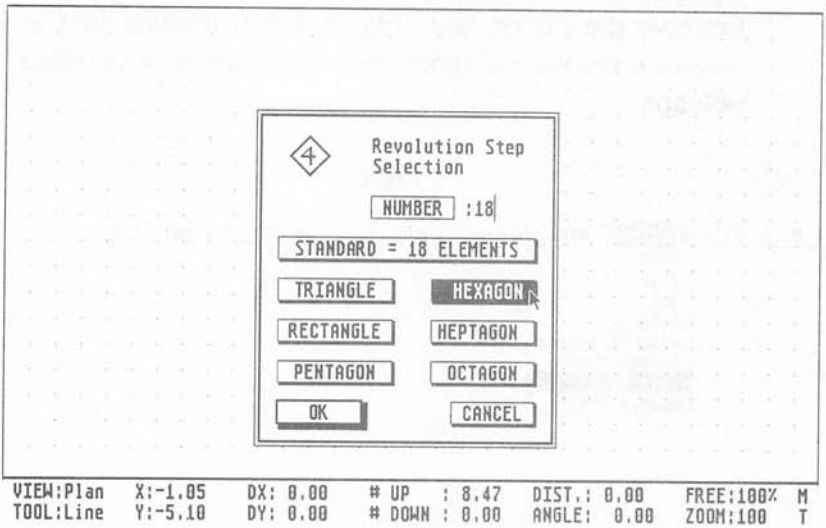
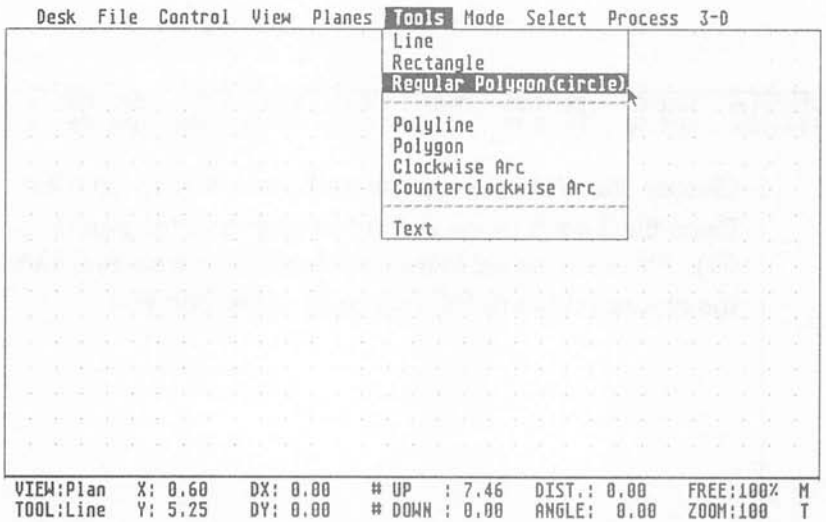
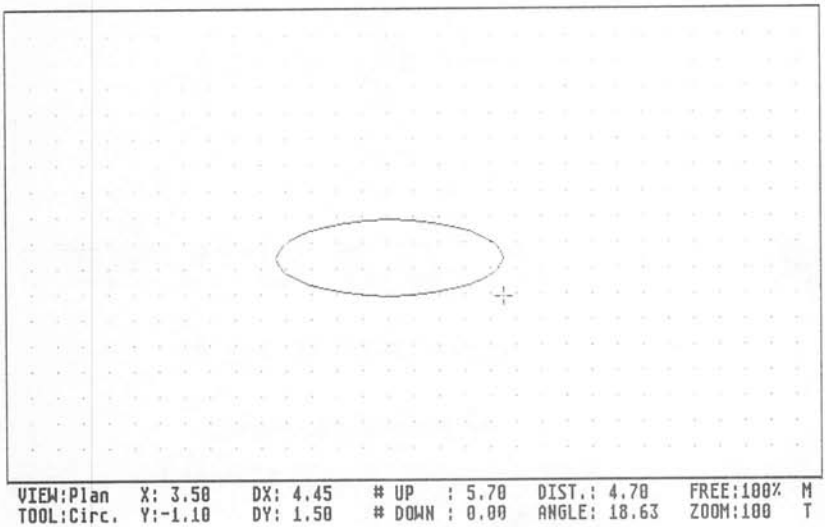

Revolution Step
Selection
NUMBER :18
STANDARD = 18 ELEMENTS
TRIANGLE
HEXAGON
RECTANGLE
HEPTAGON
PENTAGON
OCTAGON
OK
CANCEL
VIEW:Plan X:-1.05 DX: 0.00 # UP : 8.47 DIST.: 0.00 FREE:100% M
TOOL:Line Y:-5.10 DY: 0.00 # DOWN : 0.00 ANGLE: 0.00 ZOOM:100 T

Fig. 67


Desk File Control View Planes Tools Mode Select Process 3-D
Line
Rectangle
Regular Polygon(circle)
Polyline
Polygon
Clockwise Arc
Counterclockwise Arc
Text
VIEW:Plan X: 0.60 DX: 0.00 # UP : 7.46 DIST.: 0.00 FREE:100% M
TOOL:Line Y: 5.25 DY: 0.00 # DOWN : 0.00 ANGLE: 0.00 ZOOM:100 T

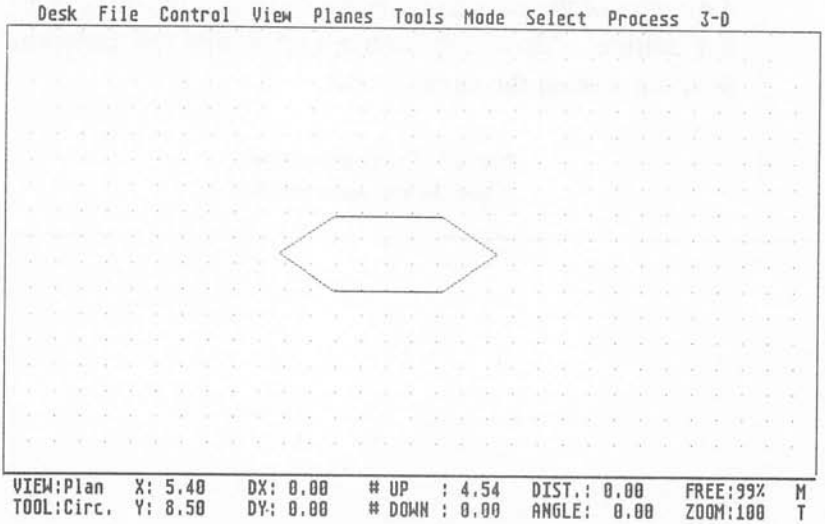
If there is already another preset object on the screen, set the center of the polygon to one side so that the objects do not collide. Move the mouse and notice the polygon forming around the center point.

Fig. 68 Using the cursor, draw the Polygon outline



Choose the form and size you wish for the polygon, and click the mouse button to set it. Notice that instead of a circle, a hexagonal polygon has appeared (Fig. 69).

Fig. 69



Execute the procedure **GENERAL VIEW...**

Fig. 70 View from the **Front...**

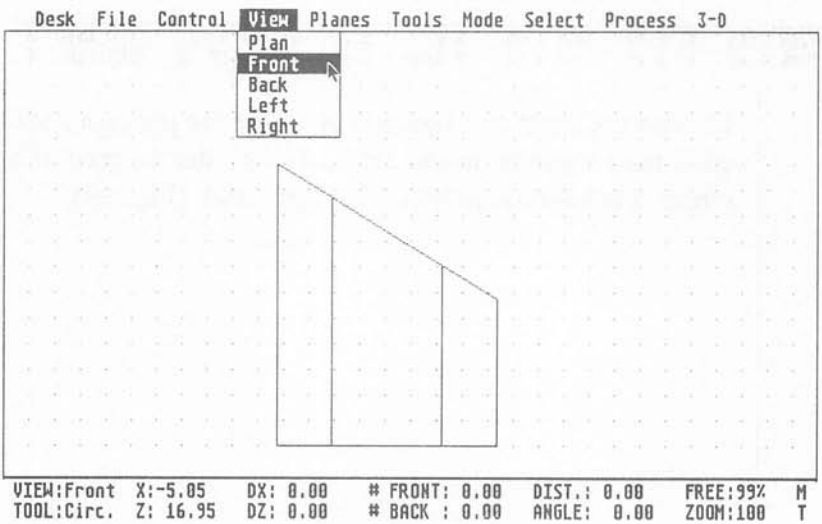


Fig. 71 ...The Left

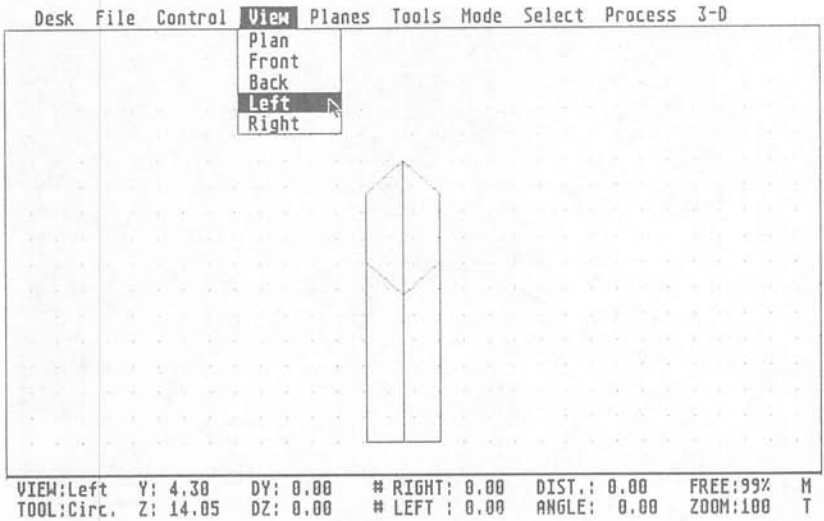


Fig. 72 Now use Filled Planes *in 3-D...*

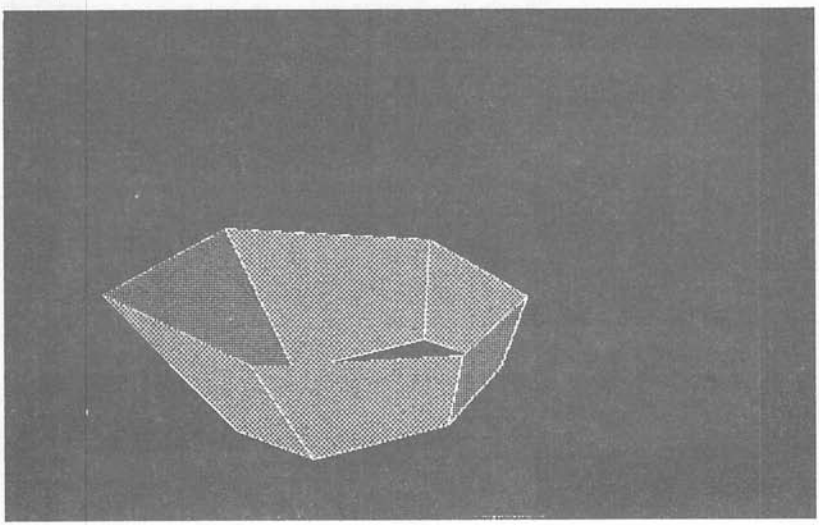


Fig. 73 ...adjust the Vertical and Horizontal View angles

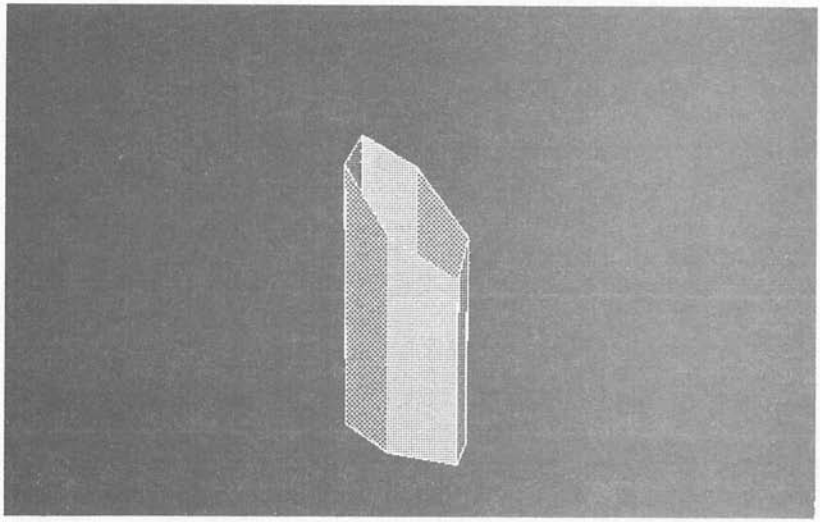
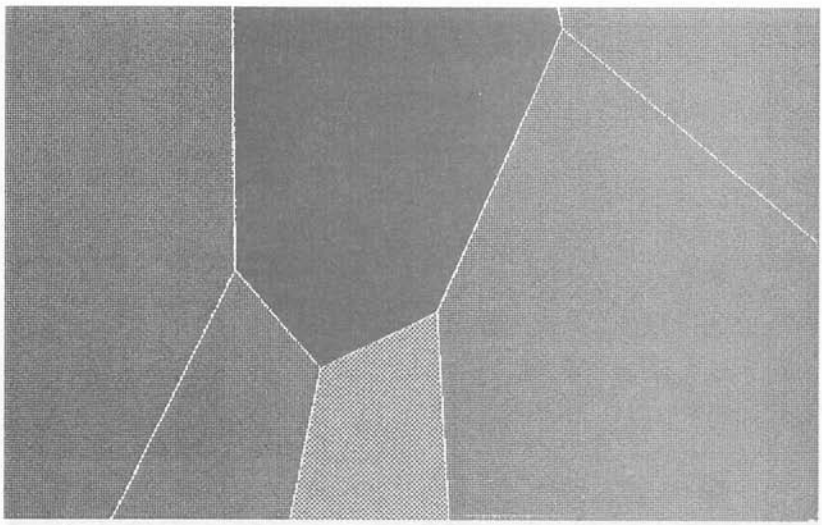


Fig. 74 View from anywhere – even inside...



Procedure : Spin

Unless you are using this procedure as part of another process, open the **File** menu and select the *New* option. If the *Attention* window appears, select *Continue* (***Warning***: This will erase anything currently on the Work Screen).

Open the **Mode** menu and select the *SPIN* option (Fig. 76). Observe in the dialog box (Fig. 77) that the option *360 SPIN* has been selected. Select *OK* to confirm the option.

Fig. 75

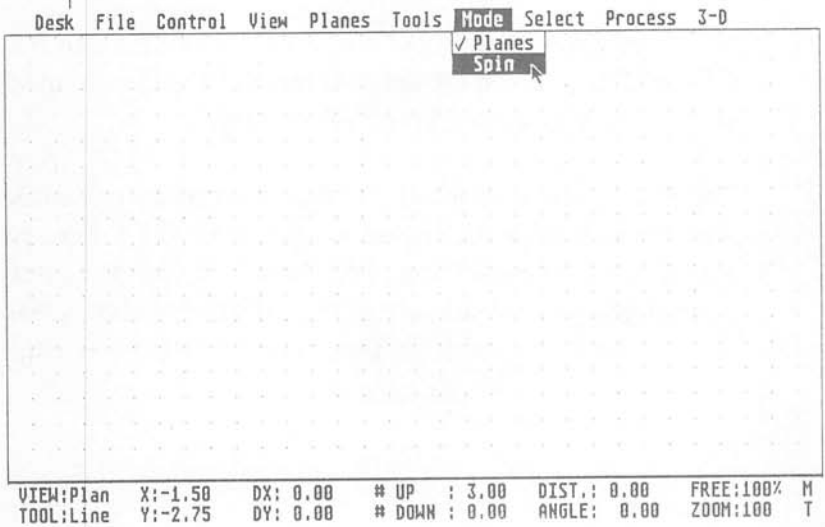
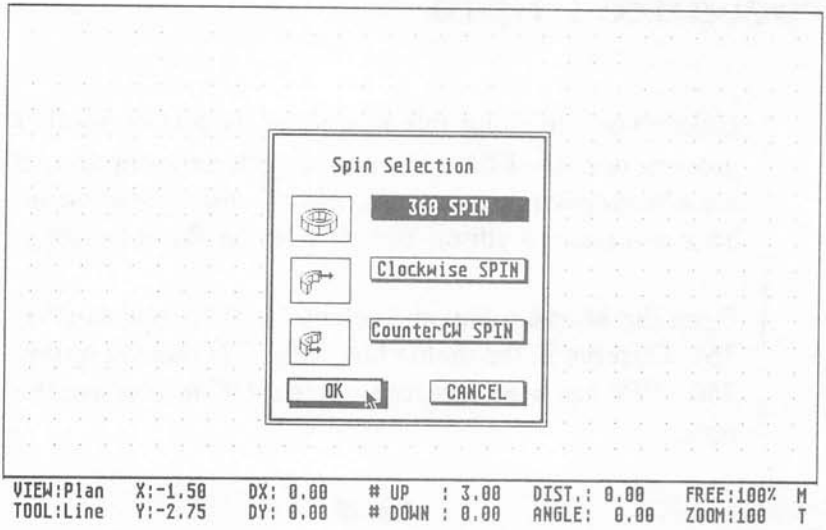


Fig. 76



Notice that the message *SET REVOLUTION CENTER BY CURSOR* appears in the upper menu bar and the monitor shows that you are working in *Plan* view.

Place the spin center as close to the center of the screen as possible and click the mouse button to set it (if there is already another preset object on the screen, set the center point slightly to one side so that the objects do not collide). Set a second point with the mouse and notice the effect.

Fig. 77 Set Revolution Center

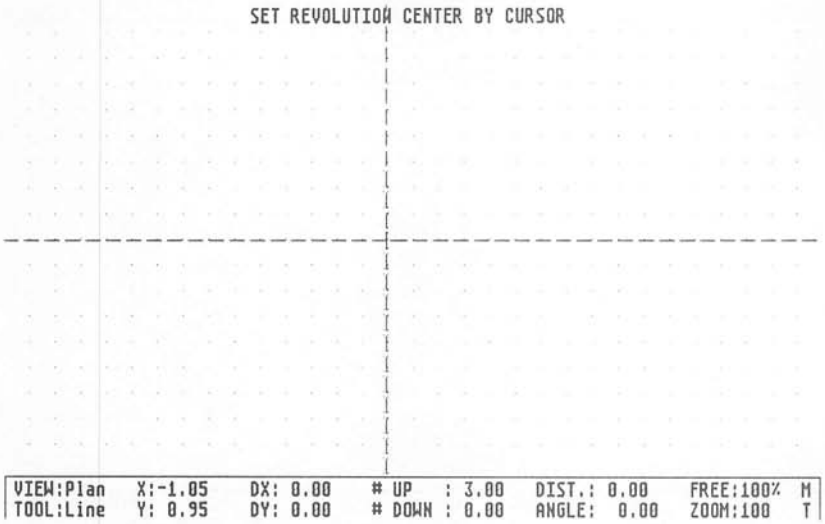


Fig. 78 Draw A Diagonal

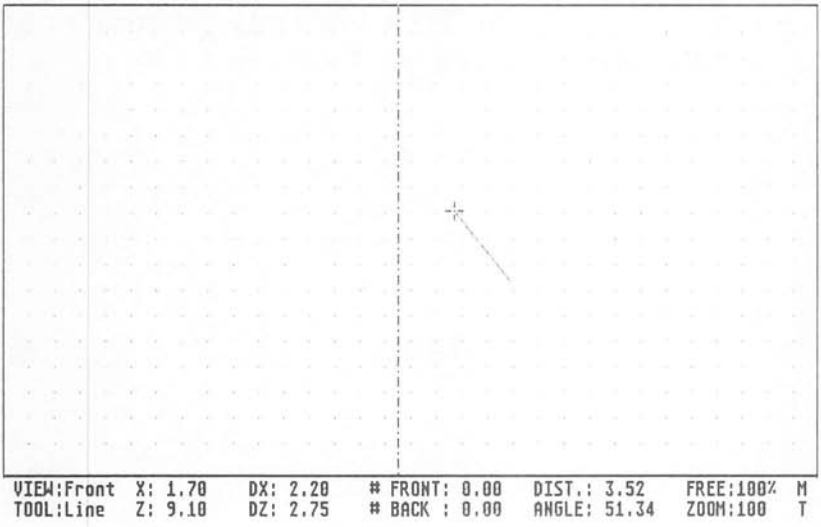
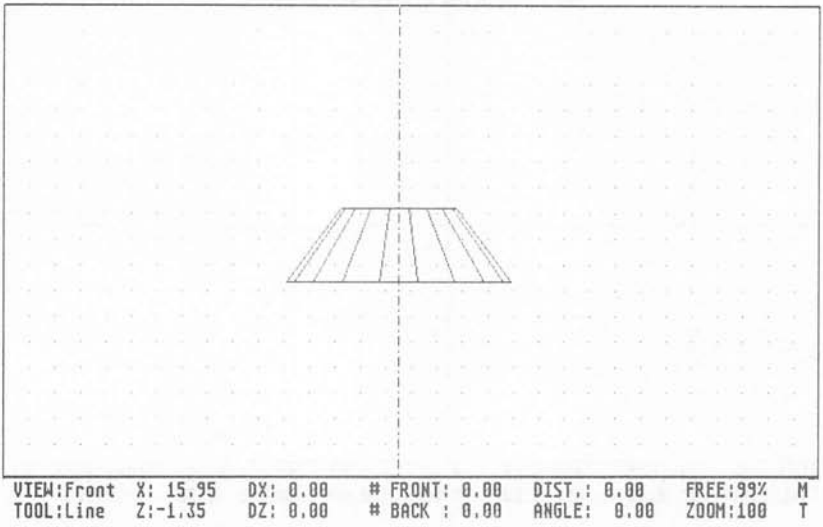


Fig. 79 Click the mouse to initiate the Spin



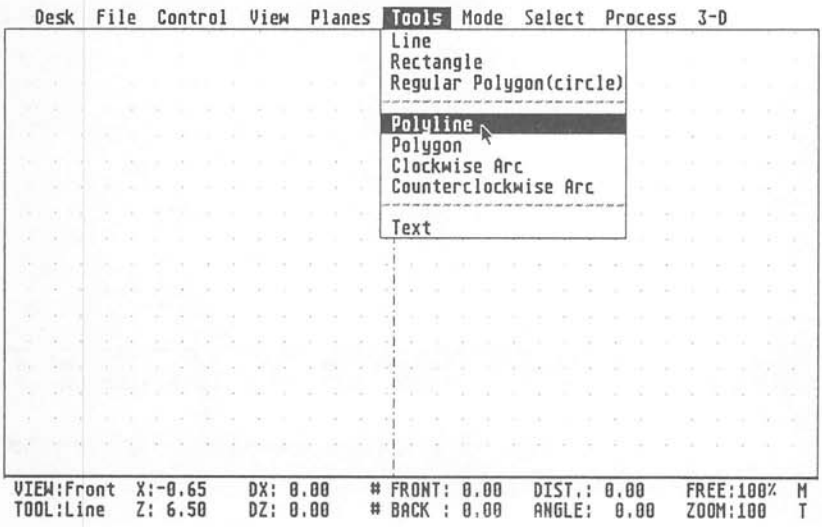
Now observe the vertical spin axis, represented by a broken line, and the *VIEW* reading in the monitor that indicates that you are in the *Front* view.

Procedure : Ring

Execute the procedure **SPIN** (See Above).

Open the **Tools** menu and select the *Polyline* option (Fig. 81).

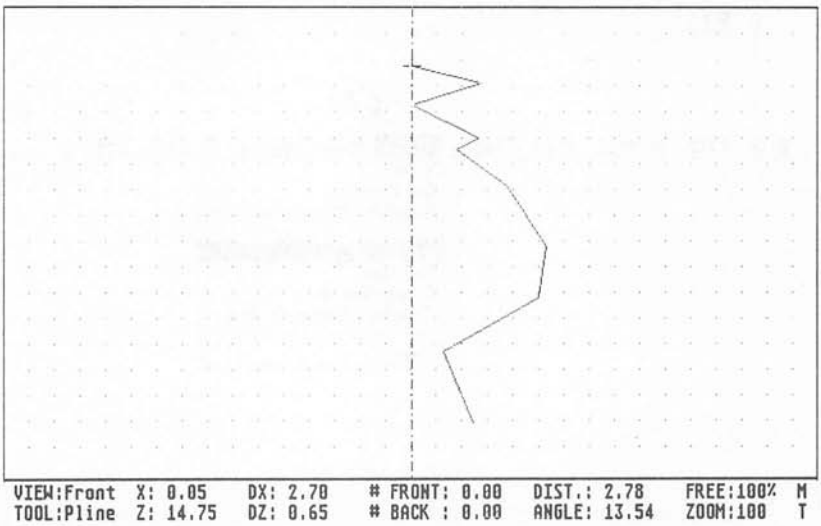
Fig. 80



Place the cursor to the right of the axis and click the mouse button to start a line.

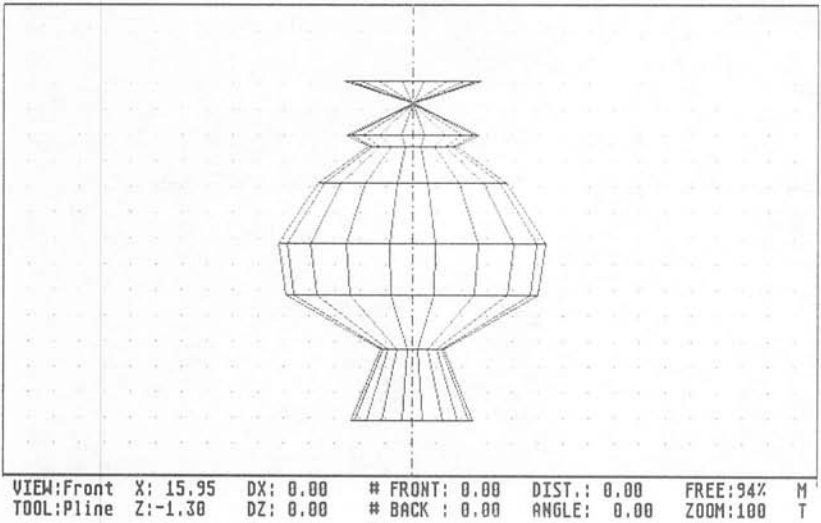
Move the mouse and define any line to the right side of the axis. Define two more lines in the same way and then press the space bar (or the right mouse button), to complete the operation.

Fig. 81 Define a Broken Line and click the right mouse button



Observe how a spun figure is generated starting from the original lines (Fig. 82)

Fig. 82 An object is formed around the selected Center of Rotation



Again Execute the **GENERAL VIEW** procedure...

Fig. 83 ...to observe the object...

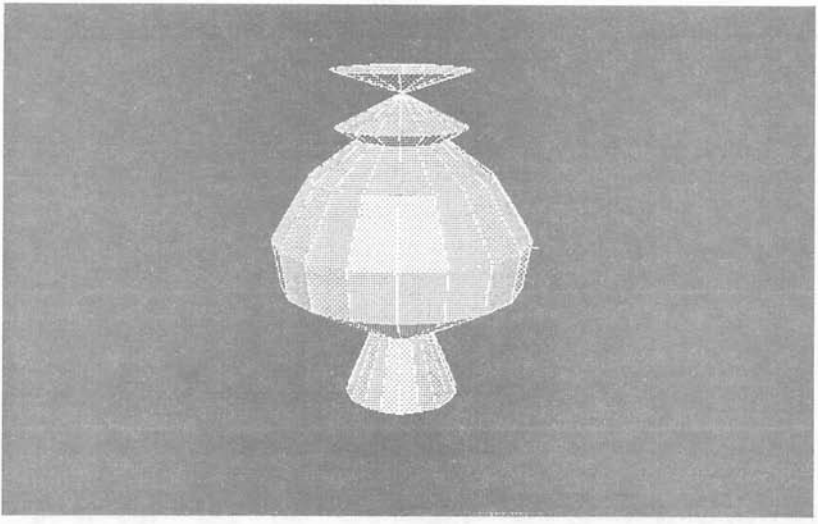


Fig. 84 ..from different elevations...

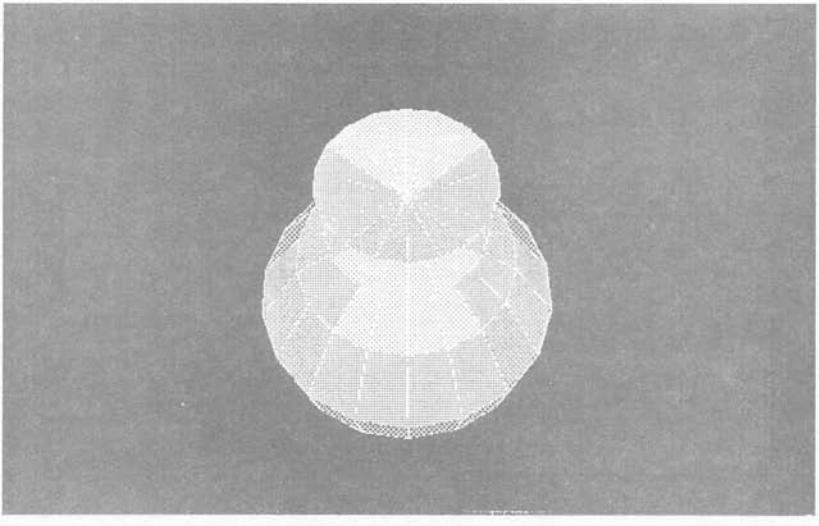


Fig. 85 ...and places
(inside looking up)

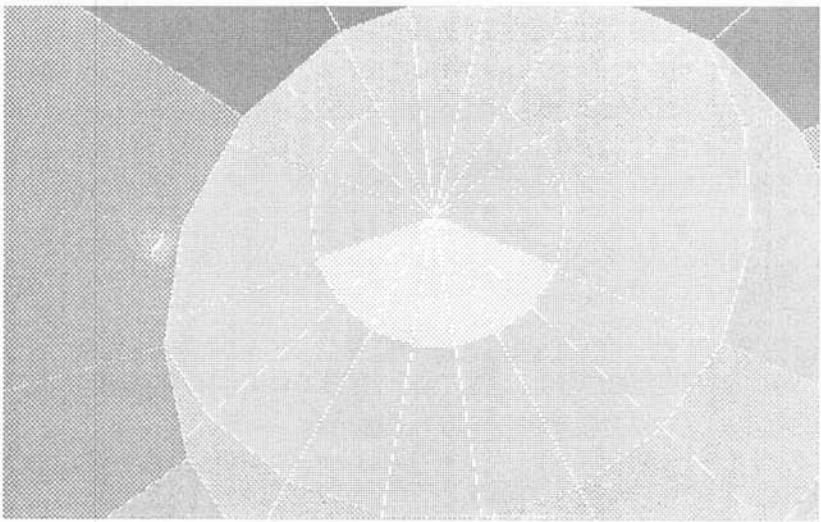
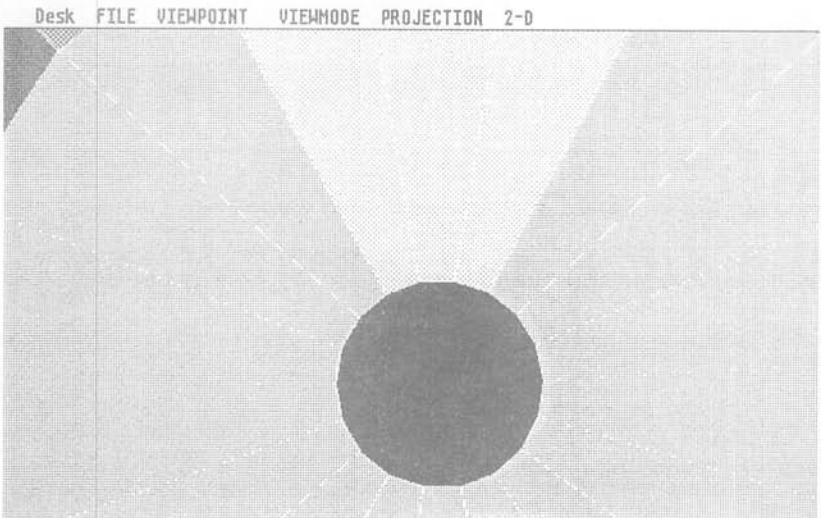


Fig. 86 (inside looking down)

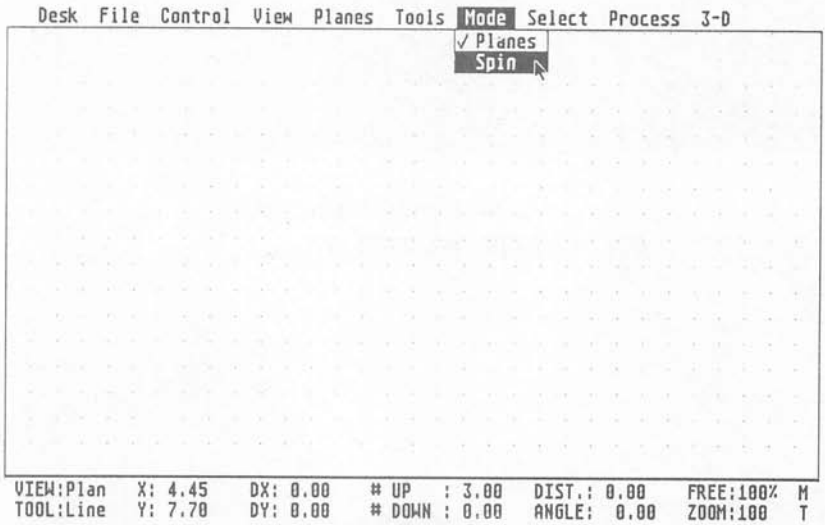


Procedure : Partial Spin

Unless you are using this procedure as part of another process, open the **File** menu and select the *New* option. If the *Attention* window appears, select *Continue* (***Warning***: This will erase anything currently on the Work Screen).

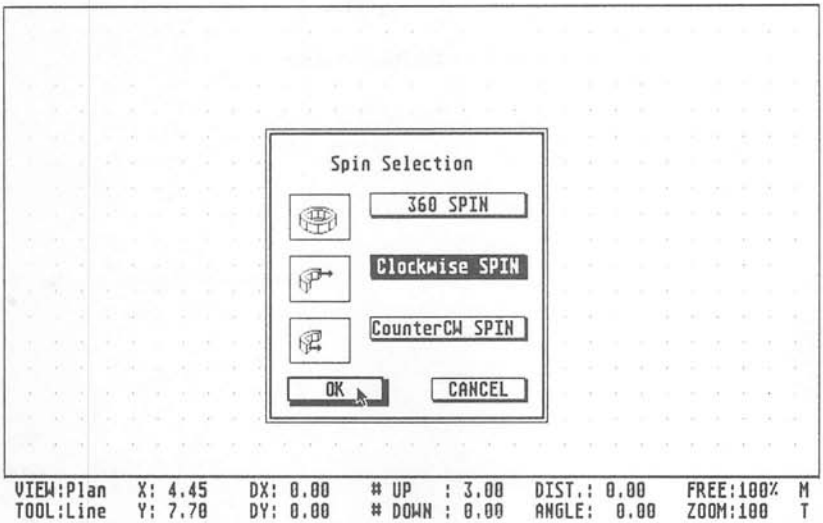
Open the **Mode** menu and select *SPIN* (Fig. 88).

Fig. 87



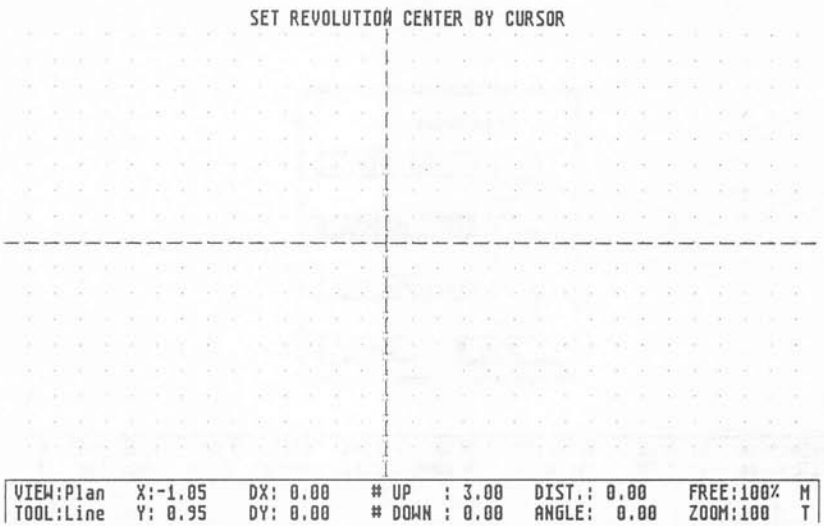
Observe in the dialog box that the *360 SPIN* option has been selected (Fig. 89). Now select the *CLOCKWISE SPIN* option and click *OK* to confirm.

Fig. 88



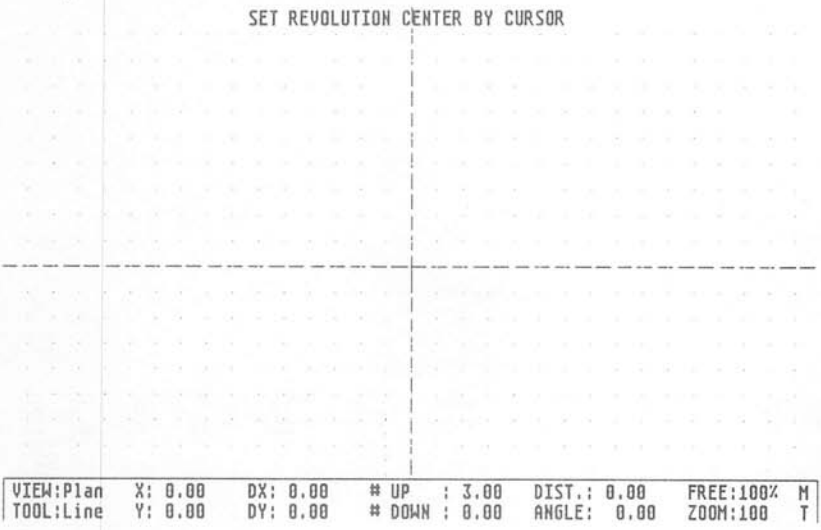
Observe the message *SET REVOLUTION CENTER BY CURSOR* on the upper menu bar and that the *View* indicator in the monitor indicates that you are in *Plan* view (Fig. 90).

Fig. 89



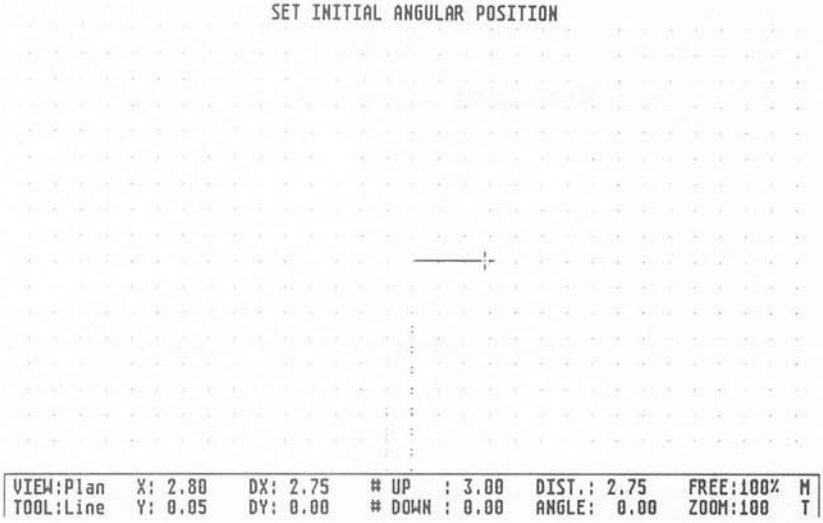
Place the center of spin as close to the center of the screen as possible and click the mouse button to set it (if there is already another preset object on the screen, set the center point slightly to one side so that the objects do not collide), set at X:0.00, Y:0.00.

Fig. 90



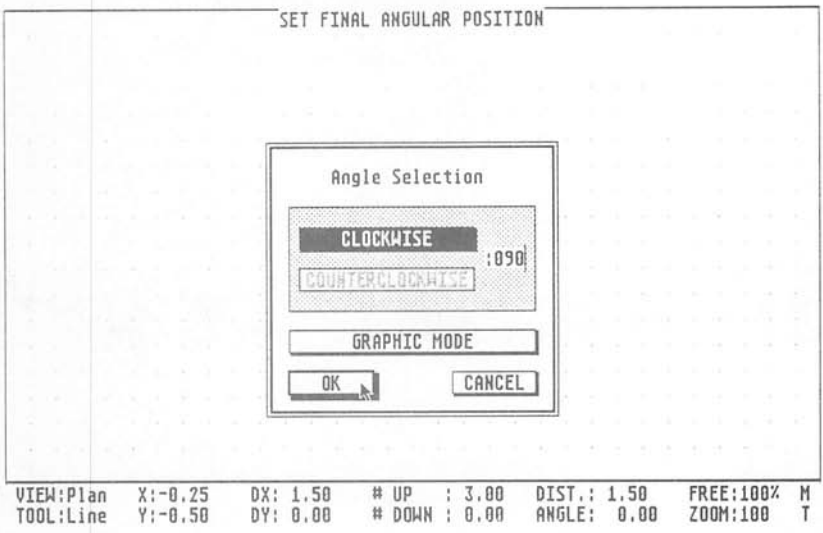
Observe the rotation indicator that pivots around the center of spin and the message *SET INITIAL ANGULAR POSITION*.

Fig. 91



Move the indicator to the right until the *Angle* readout is at 0.00 in the monitor degrees and click the mouse button to set the initial position of spin. Observe the message *SET FINAL ANGULAR POSITION*.

Fig. 92

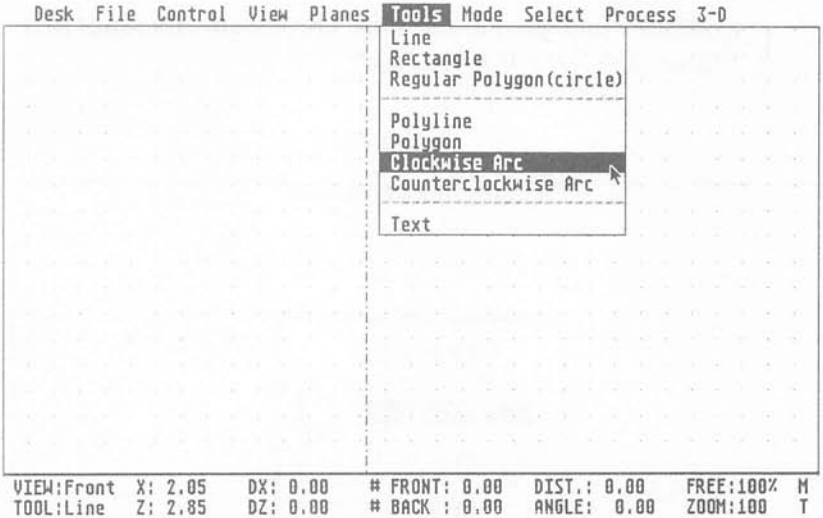


When the *Angular Selection* dialog box pops up, press Escape. The angle indicator will present a blank. Type in 90.00 degrees (one quarter turn) and click the mouse button on *OK* to set the final position of spin.

Observe now the vertical spin axis, represented by a broken line and the *View* display indicating that you are in *Front* view.

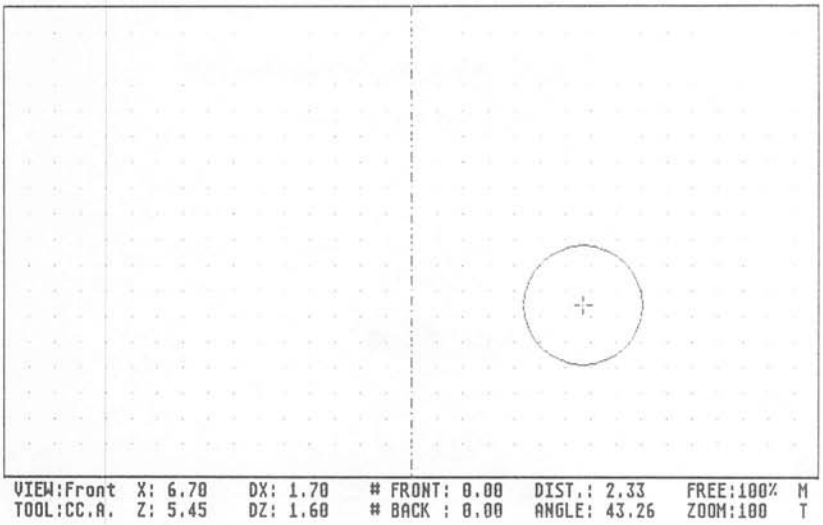
Open the **Tools** menu and select *Clockwise Arc* (Fig. 94). Place the cursor on the right side of the axis and click the mouse button to set the point of origin of the arc.

Fig. 93


Desk File Control View Planes Tools Mode Select Process 3-D
Line
Rectangle
Regular Polygon(circle)
Polyline
Polygon
Clockwise Arc
Counterclockwise Arc
Text
VIEW:Front X: 2.05 DX: 0.00 # FRONT: 0.00 DIST.: 0.00 FREE:100% M
TOOL:Line Z: 2.85 DZ: 0.00 # BACK : 0.00 ANGLE: 0.00 ZOOM:100 T

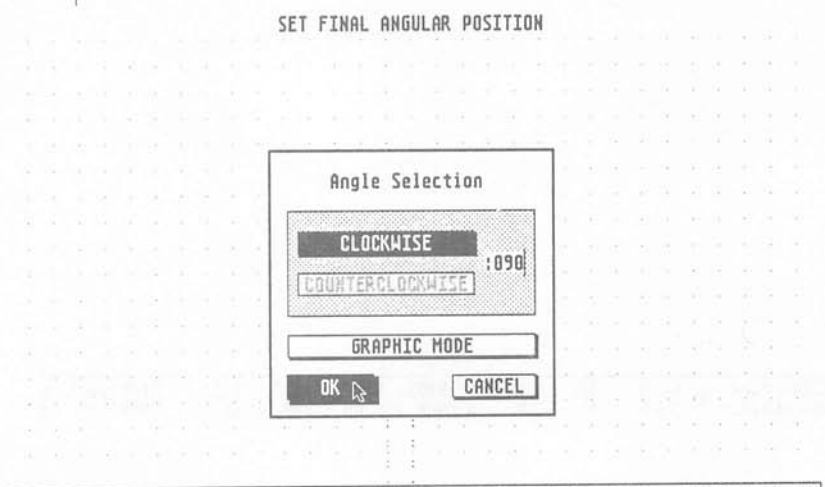
Move the cursor to the right and observe the circle that pivots around the preset point (Fig. 94).

Fig. 94



Place the second point of the circle so that it is entirely on the right side of the screen and set it by clicking the mouse button. Press escape, the angle indicator will blank, type 90.00 degrees (one quarter turn) and click the mouse button on *OK* to set the final position of spin.

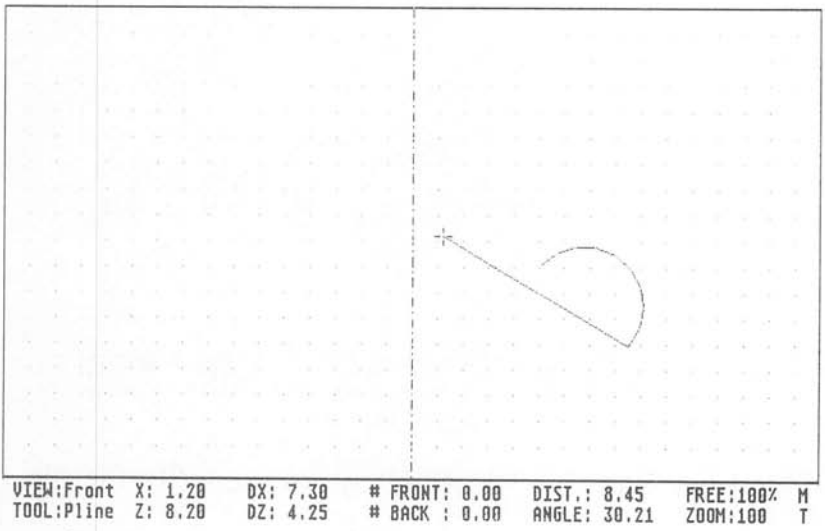
Fig. 95 Set at 90° for a quarter spin



VIEW:Plan	X: 2.80	DX: 2.75	# UP : 3.00	DIST.: 2.75	FREE:100%	M
TOOL:Line	Y: 0.05	DY: 0.00	# DOWN : 0.00	ANGLE: 0.00	ZOOM:100	T

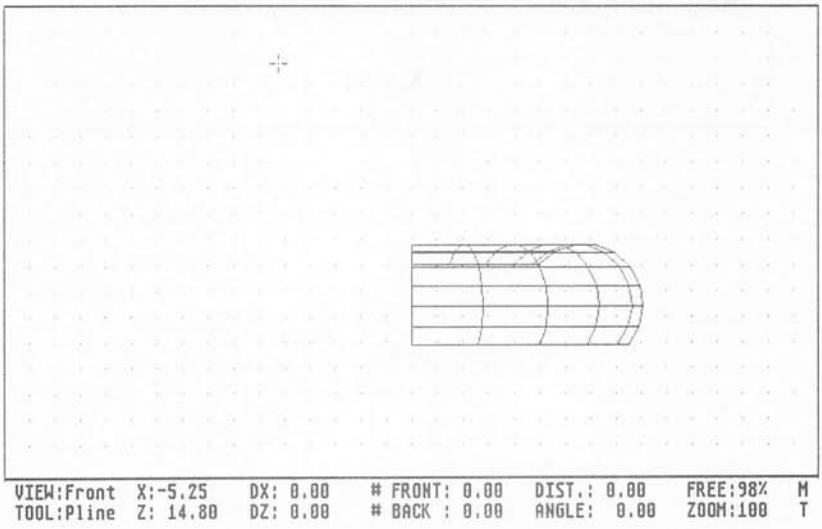
Move the mouse and observe the line that remains attached to the end of the arc and the *Tool* readout that indicates *Polyline*. Define any line (if you wish) and then press the spacebar to complete the operation (Fig. 97).

Fig. 96



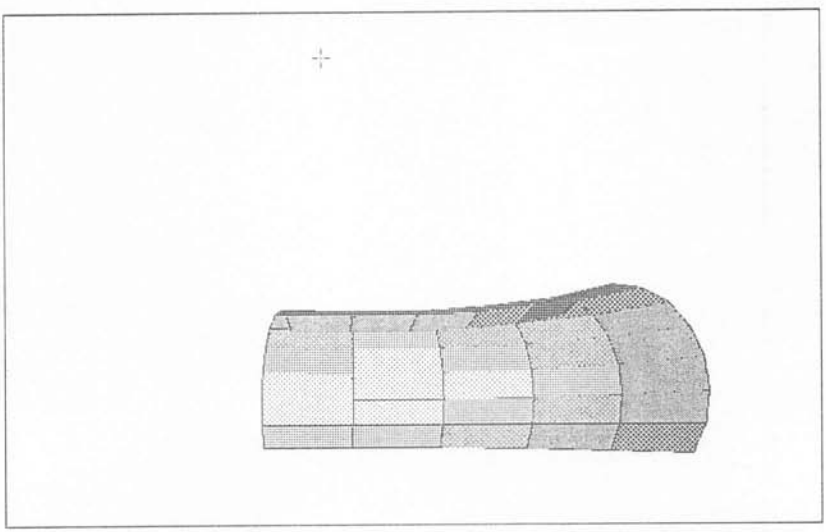
You will see the spun object being generated on the screen.

Fig. 97 The final spun object



Execute the **GENERAL VIEW** procedure (See Fig. 98).

Fig. 98

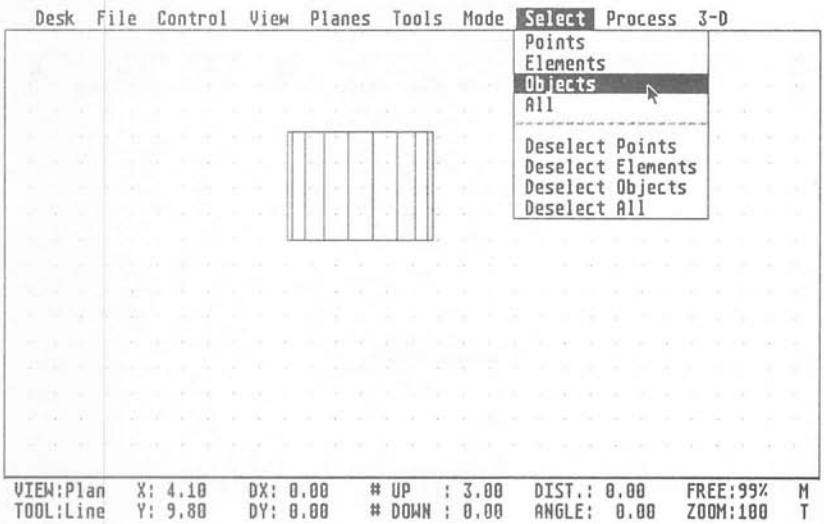


Procedure : Move Objects

Execute or open the Cylinder exercise if you saved it, or create a new one using the procedure on page 35. But do not execute **GENERAL VIEW**.

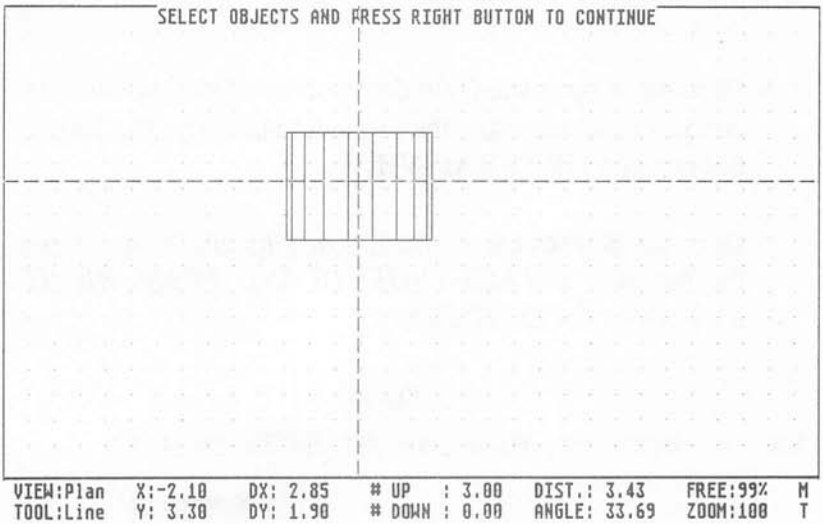
Open the **Select** menu and choose *Objects*. You will see the message *SELECT OBJECTS AND PRESS RIGHT BUTTON TO CONTINUE.*

Fig. 99



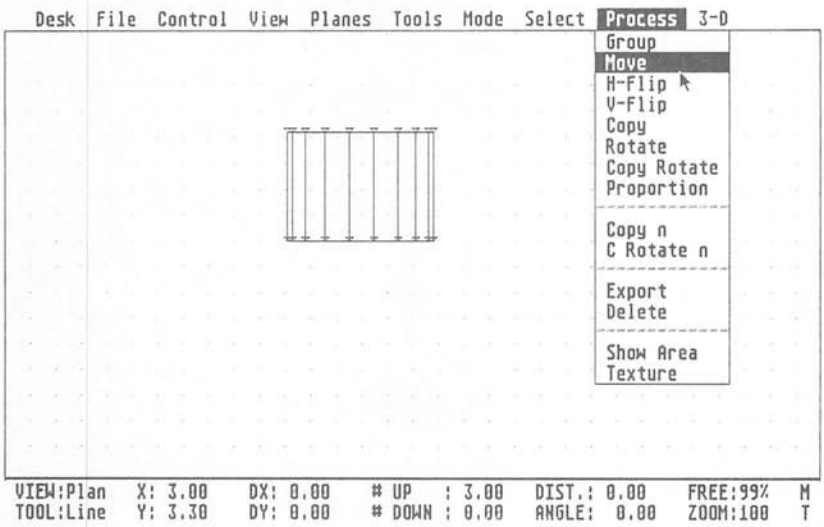
Place the cursor over one of the edges of the object and click the left mouse button to select it. Click the right mouse button to return to the drafting board.

Fig. 100 Select the Cylinder



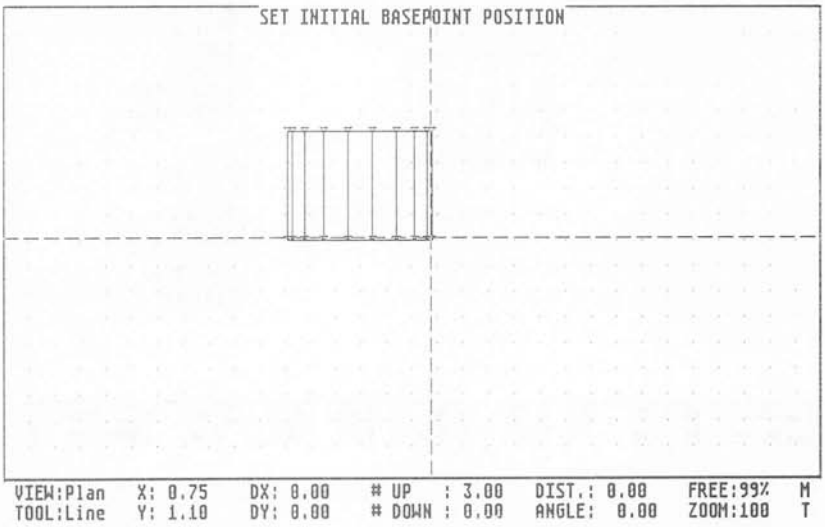
Open the **Process** menu, select *Move* and observe the message *SELECT INITIAL BASEPOINT POSITION* (Fig. 101).

Fig. 101



Now place the cursor over one of the corners of the object to mark it as a point of reference, click the mouse button to set it and observe the message *SET FINAL BASEPOINT POSITION*.

Fig. 102 Select the Initial Basepoint position



Now move the cursor to any position on the screen and click the mouse button to set it. Notice how the object moves to the new position.

Fig. 103 Set the Final Basepoint position...

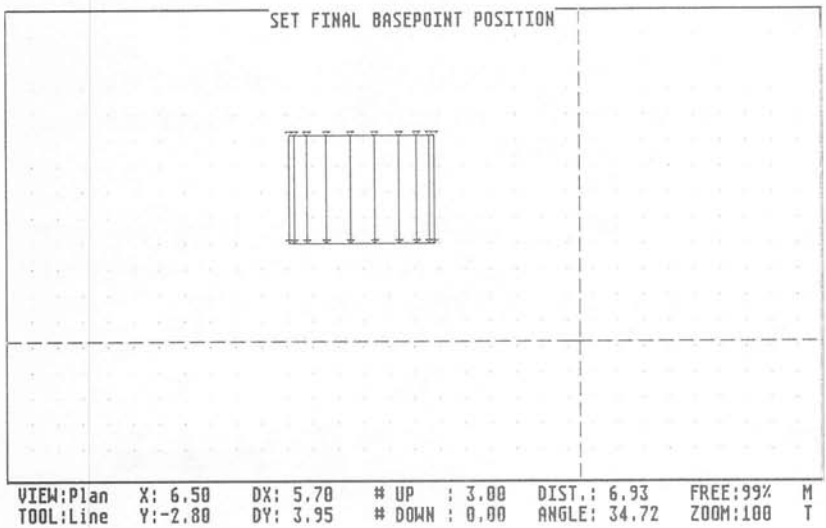
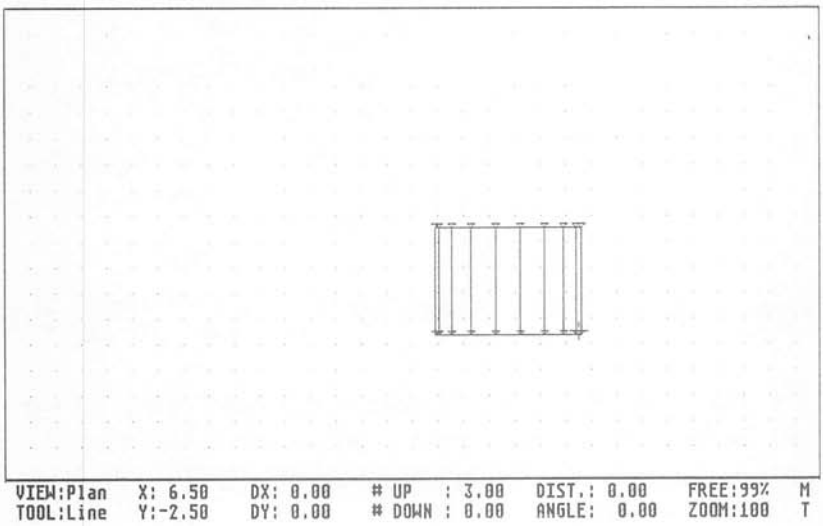


Fig. 104 ...and click to move

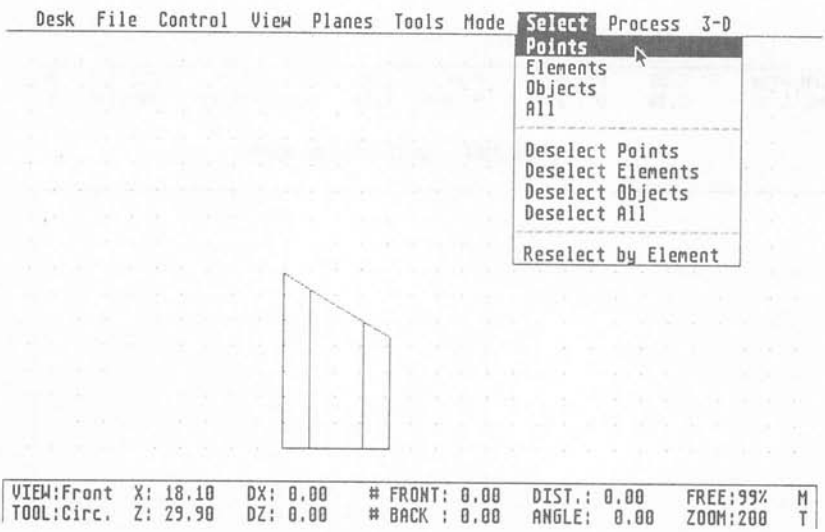


Procedure : Flip Elements

Open the HEXAGONAL PRISM created earlier or execute the procedure on page 57, but without executing **GENERAL VIEW**.

Open the **Select** menu and choose POINTS (Fig. 105). Observe the message *SELECT POINTS AND PRESS THE RIGHT MOUSE BUTTON TO CONTINUE.*

Fig. 105



Place the cursor over the upper left-hand corner of the hexagon and click the left mouse to select it. Now trace the outline box completely around the object. Click the right mouse button to confirm.

Fig. 106 Place cursor and click left mouse button...

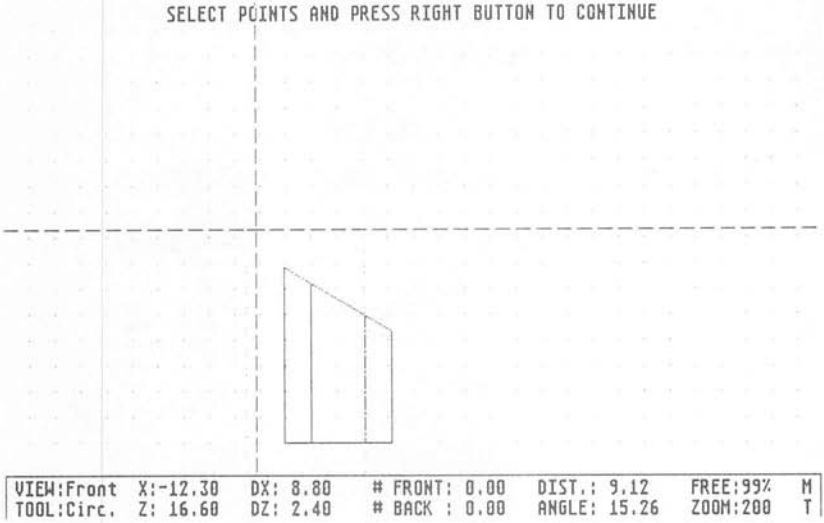
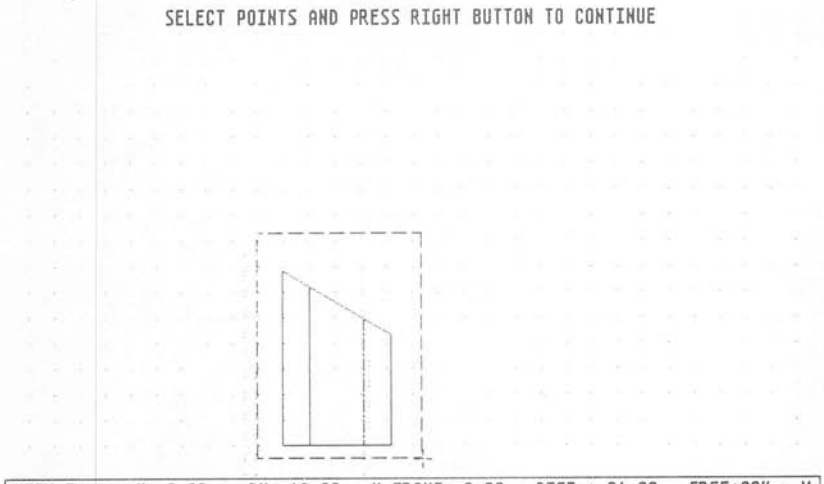


Fig. 107 Move down and right to form a selection box...



Open the **Process** menu and select *H-Flip* (Horizontal Flip) (Fig. 108). Observe the axis on the screen. Move it until it touches the extreme right point of the hexagon and click the mouse button to define it.

Fig. 108

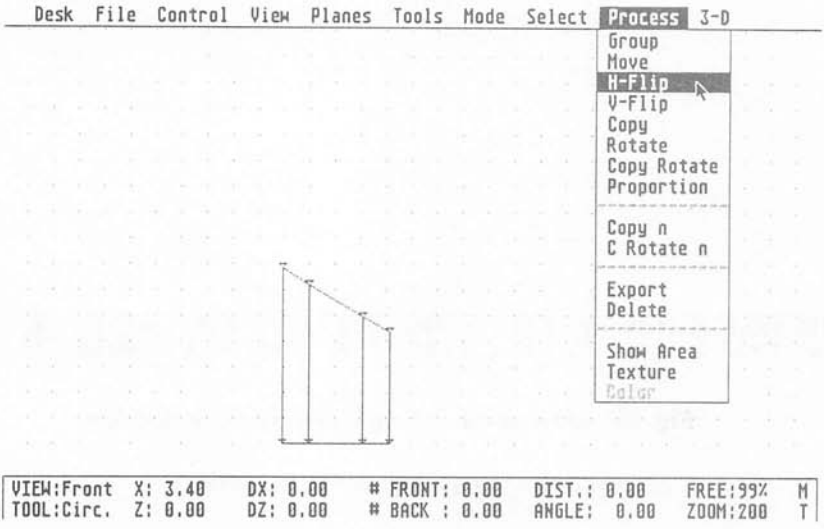
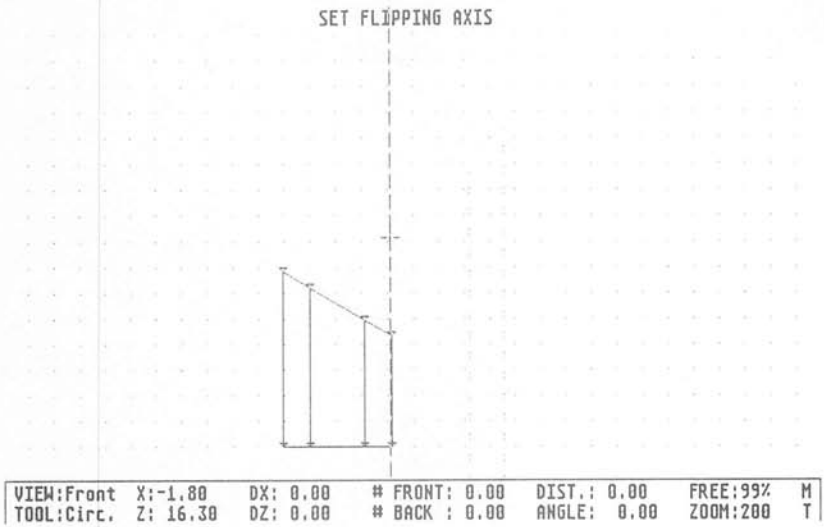
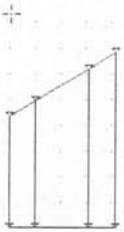


Fig. 109 Move the axis to the Hexagon and click



Observe the new position and orientation of the element (Fig. 110).

Fig. 110



```
VIEW:Front  X:-1.90    DX: 0.00    # FRONT: 0.00   DIST.: 0.00    FREE:99%   M
TOOL:Circ.  Z: 16.60   DZ: 0.00    # BACK : 0.00   ANGLE:  0.00   ZOOM:200   T
```

Open the **Process** menu again and select V-FLIP (Vertical Flip) (Fig. 111), and observe the axis on the screen (Fig. 112). Move it until it touches the extreme top point of the hexagon and click the mouse button to define it (Fig. 113).

Fig. 111

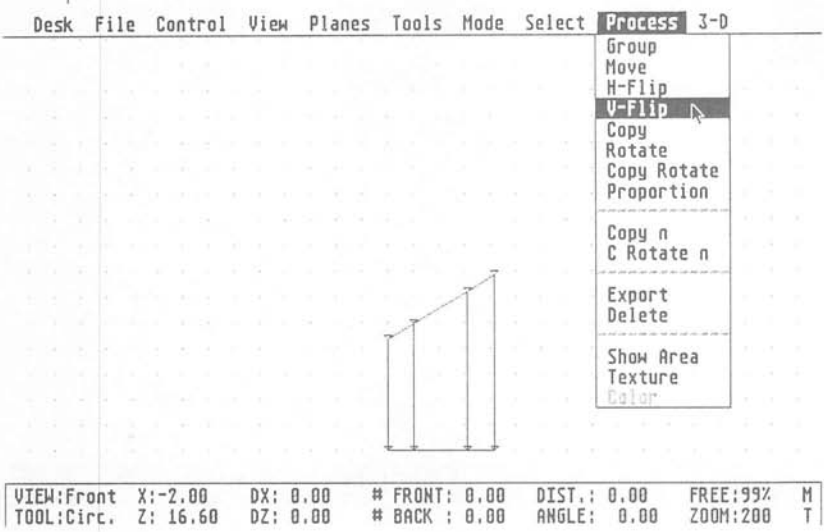


Fig. 112

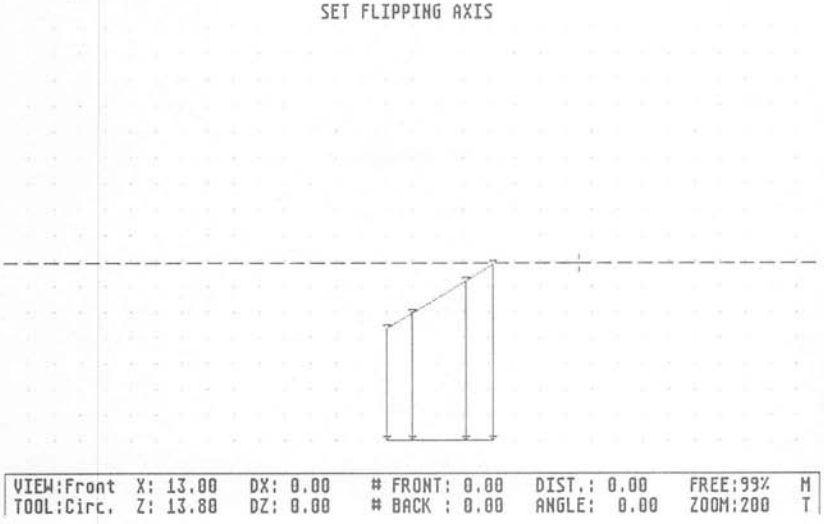
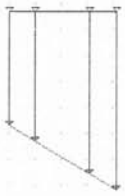


Fig. 113



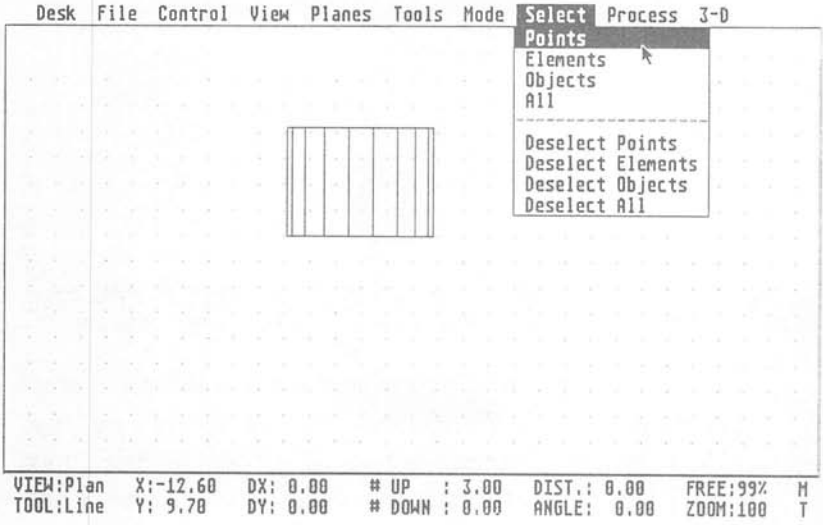
VIEW:Front	X: 13.00	DX: 0.00	# FRONT: 0.00	DIST.: 0.00	FREE:99%	M
TOOL:Circ.	Z: 13.50	DZ: 0.00	# BACK : 0.00	ANGLE: 0.00	ZOOM:200	T

Execute the **GENERAL VIEW** procedure if you wish.

Procedure : Copy

Open the CYLINDER.OBJ created earlier, or execute the procedure described on page 35. Do not execute **GENERAL VIEW**.

Open the **Select** menu and choose *Points* (Fig. 114). Observe the message *SELECT POINTS AND PRESS RIGHT BUTTON TO CONTINUE*.

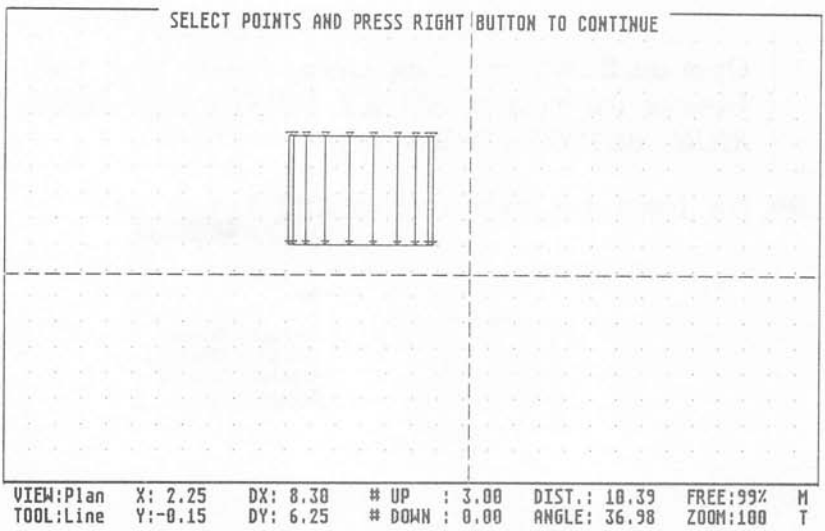


To select the points to copy, it is necessary to define a rectangular area that will contain them. To do this, place the cursor at some point on the upper left-hand part of the screen, above the circle, and click the mouse button to initialize the area.

Move the mouse to the right, and notice the rectangle drawn with broken lines that appears attached to the preset point and to the cursor. Define a rectangular area that

contains the entire circle and click the mouse button to select it. Observe the selection indicators that are now part of the circle. (If for some reason some point is not indicated, repeat the previous procedure until all points are selected.)

Fig. 115



Press the right mouse button to complete the process of selection. Open the **Process** menu and select *Copy* (Fig. 116). Observe the message *SET INITIAL BASEPOINT POSITION*. Place the cursor in the center of the circle and click the mouse button to mark it as a point of reference.

Fig. 116 Select Copy

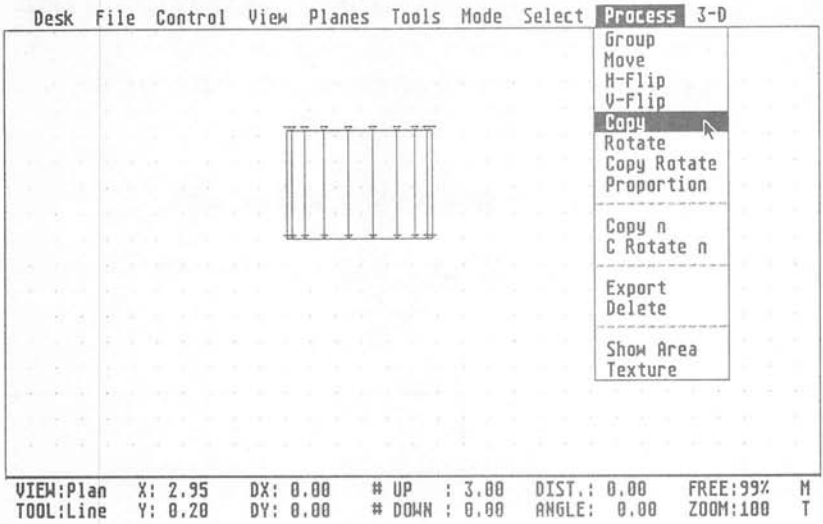
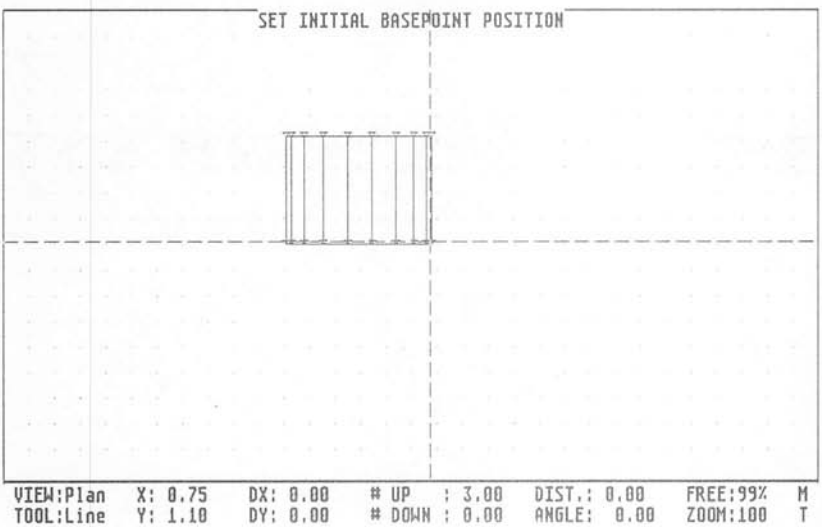


Fig. 117 Set the Initial Basepoint



Observe the message *SET FINAL BASEPOINT POSITION*. Move the cursor to any point outside the circle and click the mouse button to define it as the center of the new circle.

Fig. 118 Set Final Basepoint and click...

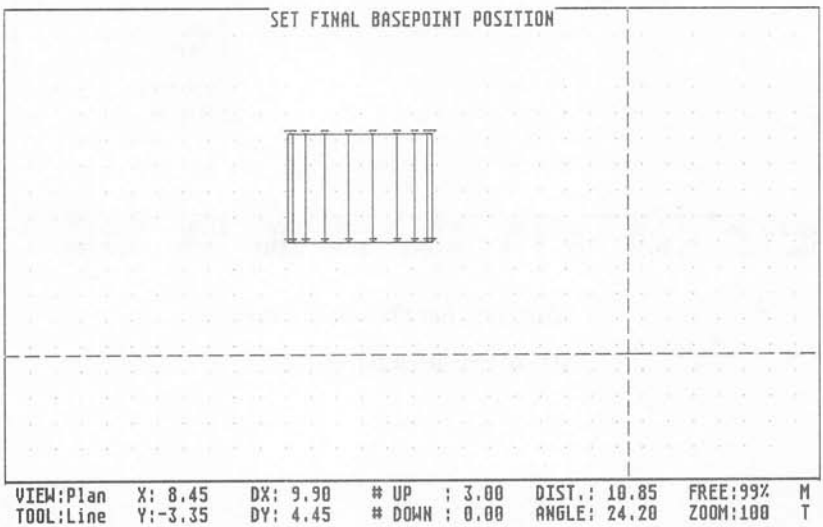
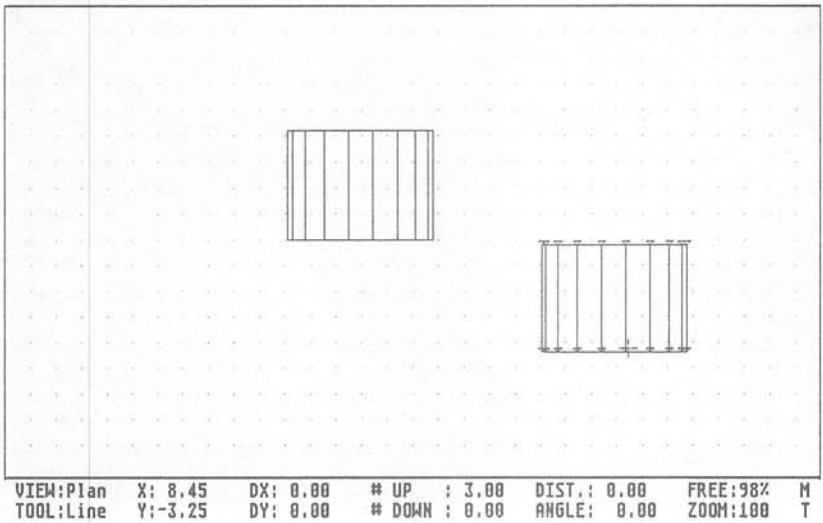


Fig. 119 Copy of object appears at set point



GENERAL VIEW may be executed at this point if desired (See Figures 120-125).

Fig. 120

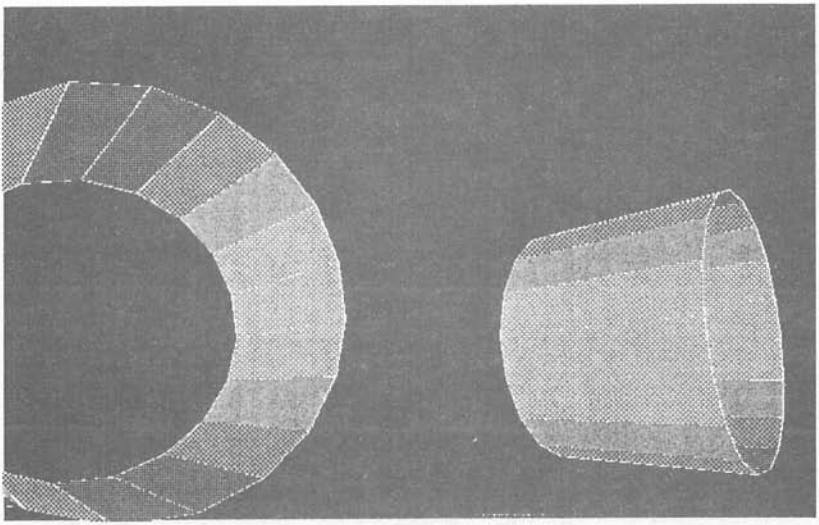


Fig. 121

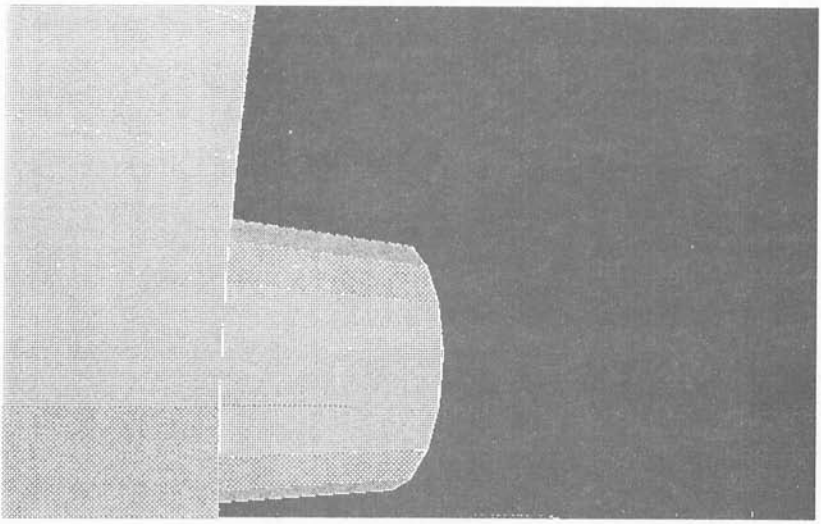


Fig. 122

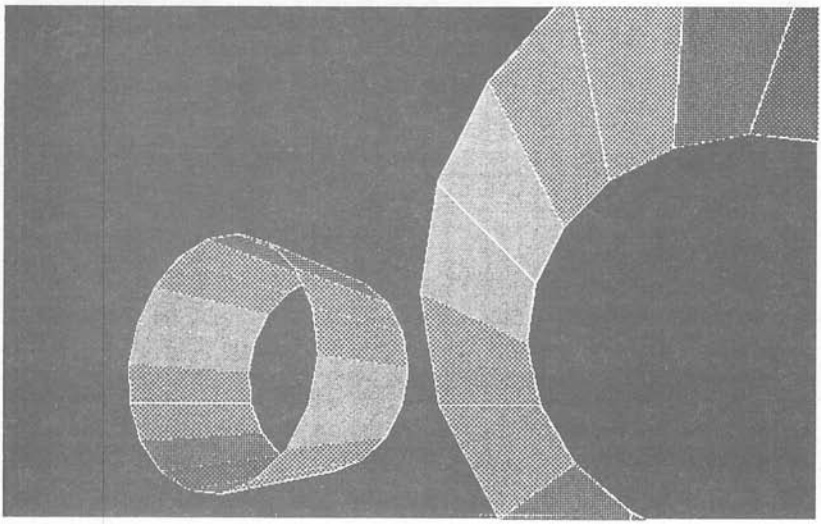


Fig. 123

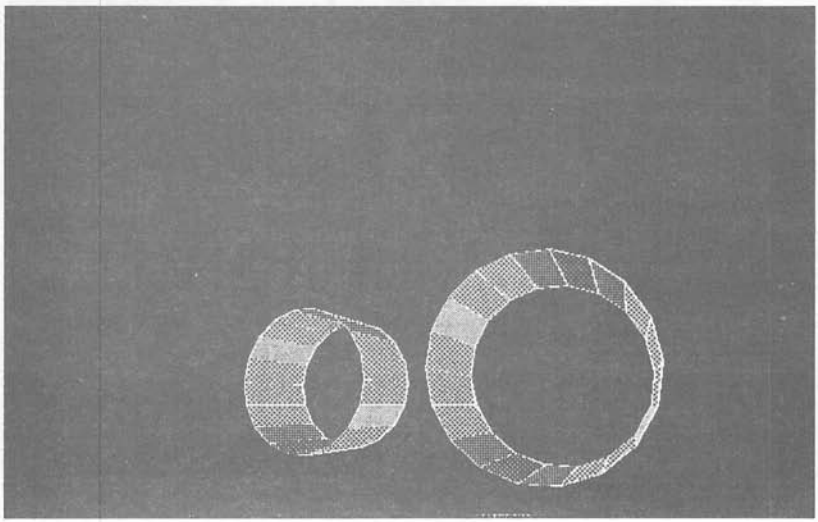


Fig. 124

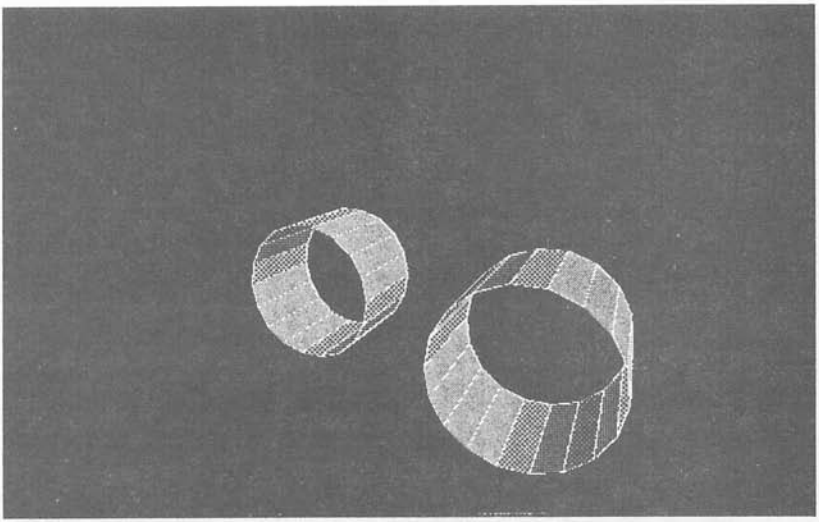
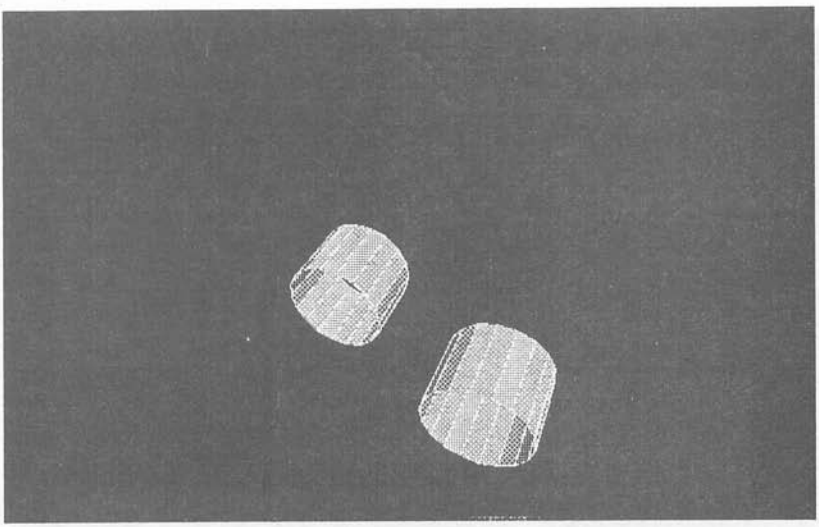


Fig. 125

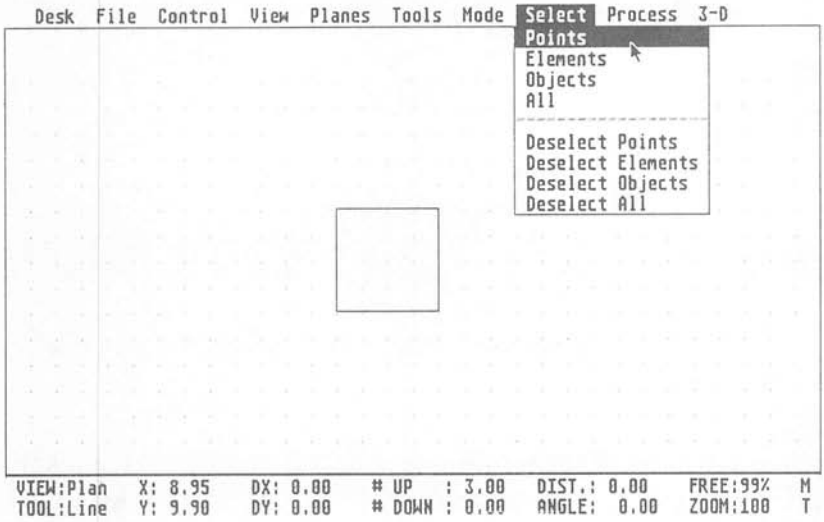


Procedure : Rotate Elements

Open the CUBE.OBJ created earlier or execute the CUBE procedure without executing **GENERAL VIEW**.

Open the **Select** menu and choose *Points* (Fig. 126). Observe the message *SELECT POINTS AND PRESS RIGHT BUTTON TO CONTINUE*.

Fig. 126

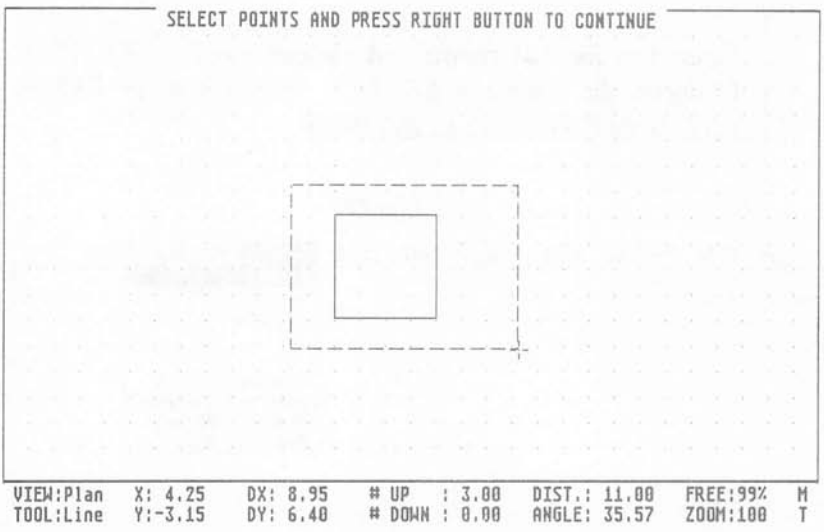


To select the points to rotate, it is necessary to define a rectangular area that will contain them. To do this, place the cursor at some point on the upper left-hand part of the screen, above the cube, and click the mouse button to initialize the area.

Move the mouse to the right, and notice the rectangle drawn with broken lines that appears attached to the preset point and to the cursor. Define a rectangular area that

contains the entire cube and click the mouse button to select it. Observe the selection indicators that are now part of the cube (if for some reason some point is not indicated, repeat the previous procedure until all points are selected).

Fig. 127 Use the rectangle to select the cube



Press the right mouse button to finalize the process of selection.

Open the **Process** menu, select *Rotate* (Fig. 128), and observe the message *SET SPIN CENTER* (Fig. 129). Place the cursor over one of the sides of the chosen element and click the mouse button to set the center of rotation.

Fig. 128

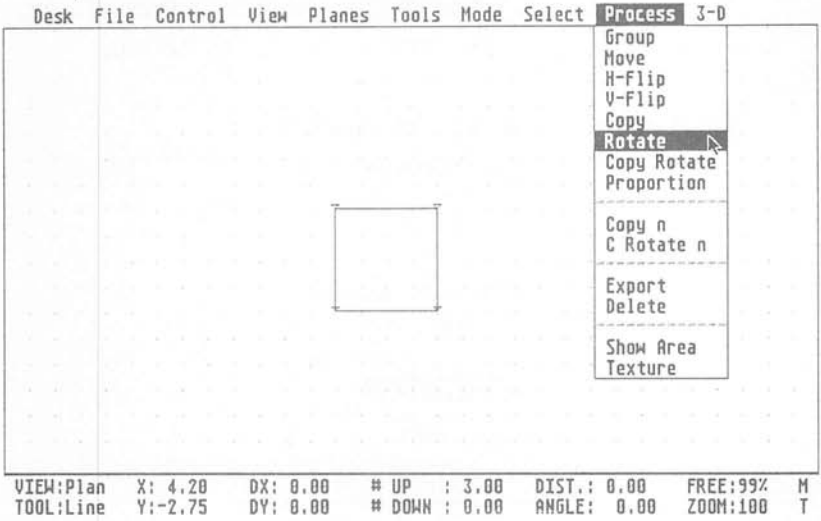
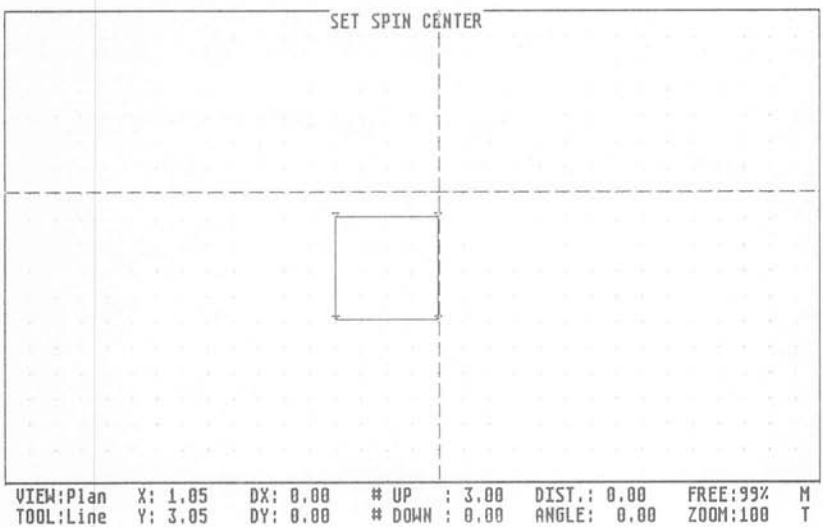
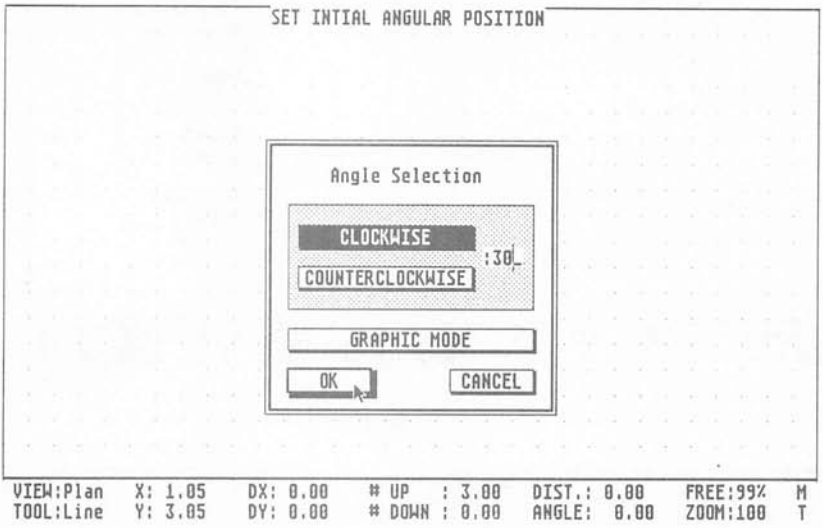


Fig. 129



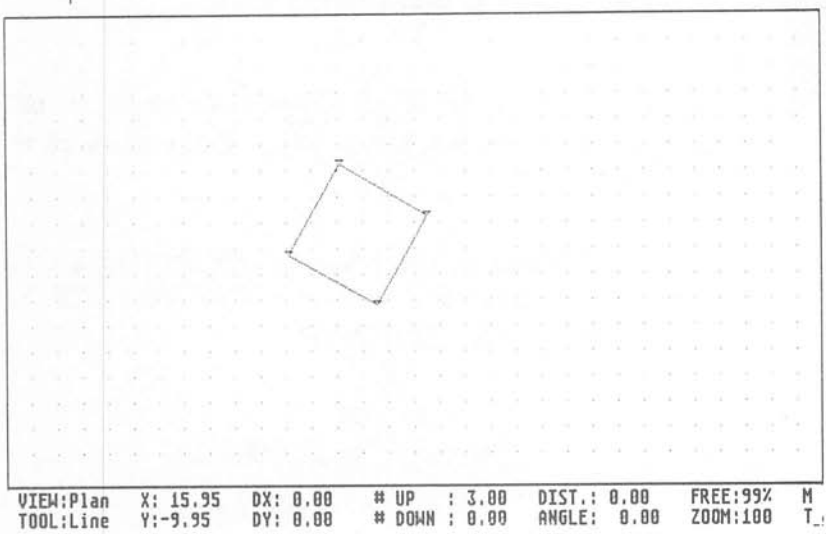
The *Angle Selection* dialog box will appear. Press escape to clear the three 0's from the degree indicator and insert 30, with clockwise selected. Press the *OK* button.

Fig. 130 Set **Angle** *at 30*



The object (cube) has now rotated 30° to the right of its original position.

Fig. 131

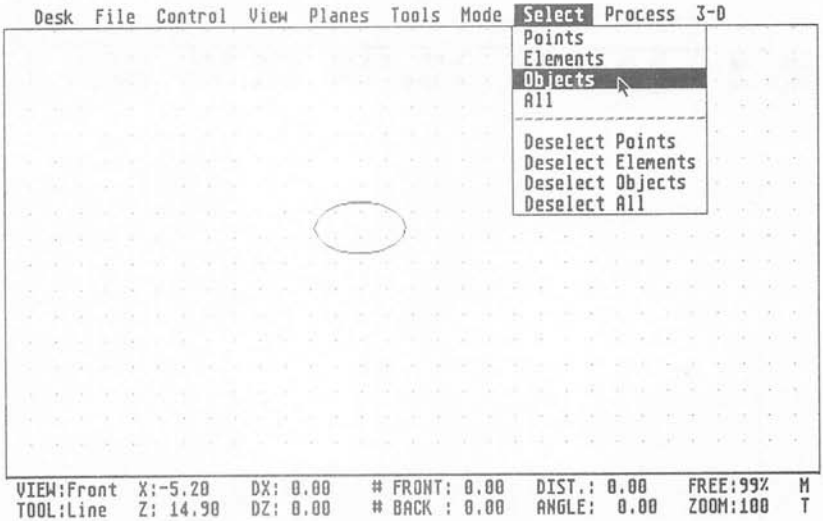

VIEW:Plan X: 15.95 DX: 0.00 # UP : 3.00 DIST.: 0.00 FREE:99% M
TOOL:Line Y:-9.95 DY: 0.00 # DOWN : 0.00 ANGLE: 0.00 ZOOM:100 T_

Procedure : Proportion

Execute the **CYLINDER** procedure from page 35, or open the one executed previously. Do not execute **GENERAL VIEW.**

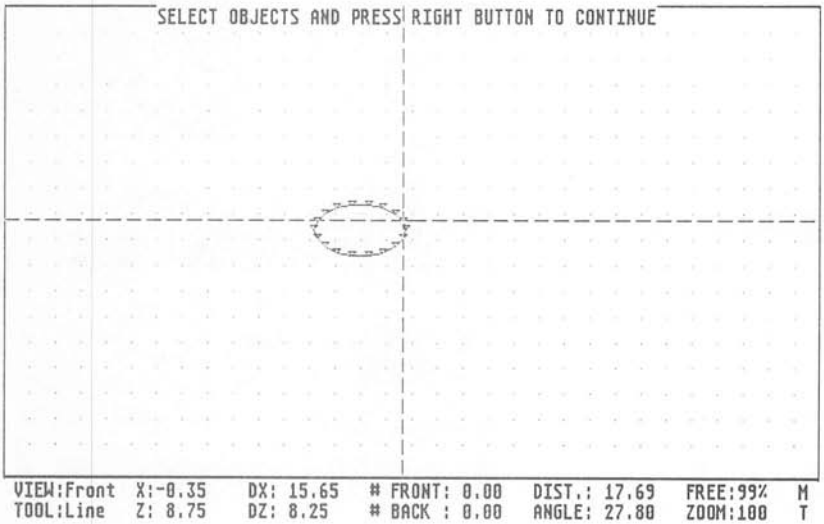
Open the **Select** menu and choose OBJECTS (Fig. 132). Observe the message *SELECT OBJECTS AND PRESS RIGHT BUTTON TO CONTINUE.*

Fig. 132



Place the cursor over any point on the perimeter of the polygon and click the mouse button to select it. Observe the message *SELECT OBJECTS AND PRESS RIGHT BUTTON TO CONTINUE*. Move the cursor to the side of the cylinder and press the right mouse button to select (Fig. 133).

Fig. 133



Open the ***Process*** *menu* and select *Proportion*. Observe the message *SET PROPORTION CENTER*.

Fig. 134 Select Proportion from the PROCESS *Menu*

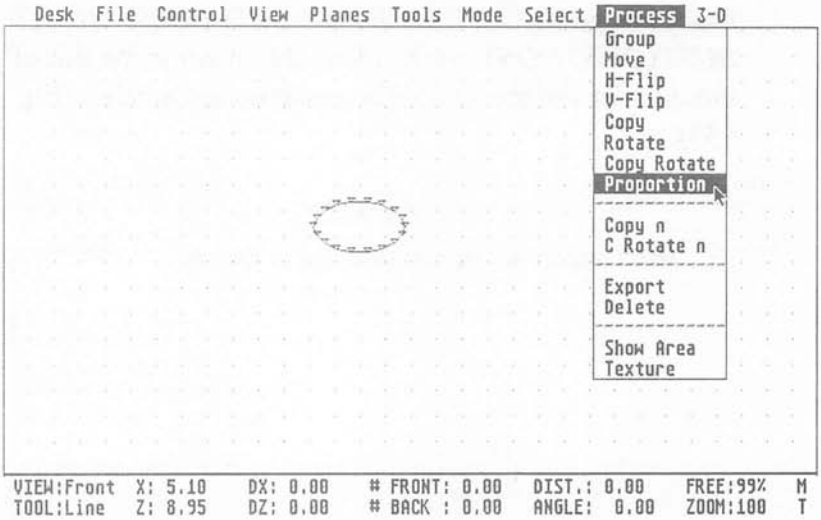
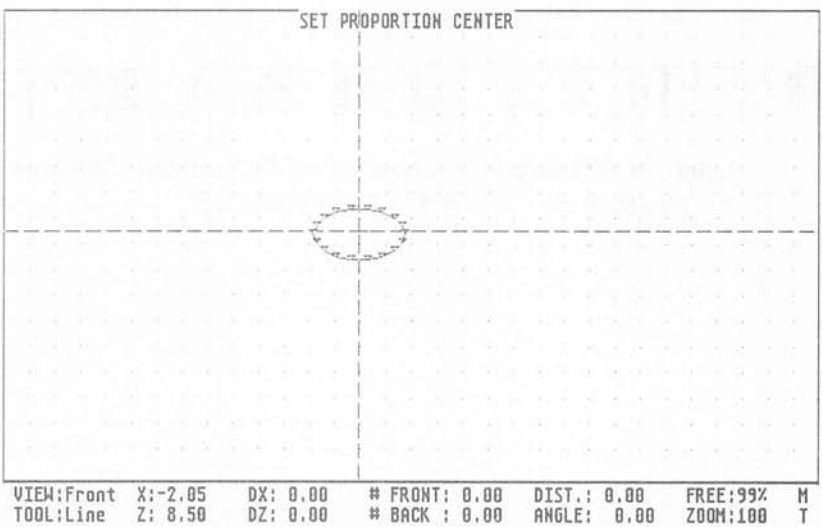
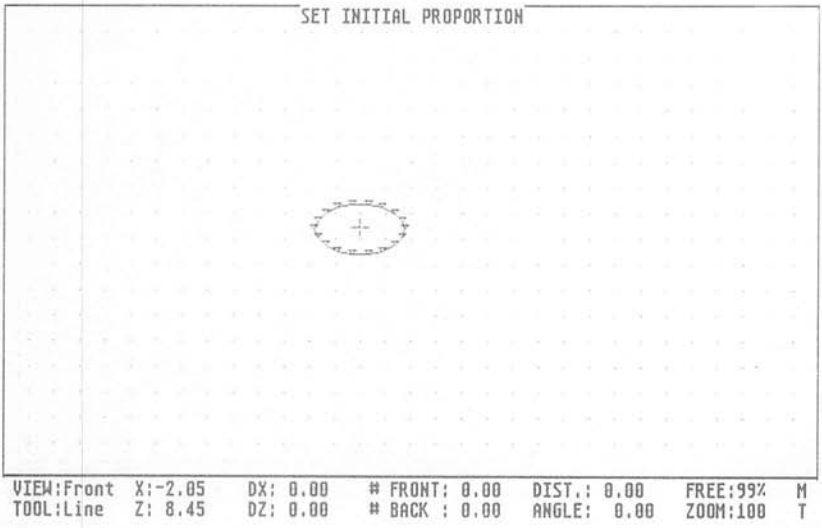


Fig. 135 Place the cursor at the Center...



Place the cursor in the center of the polygon and click the mouse button to define it. Notice the message *SET INITIAL PROPORTION*.

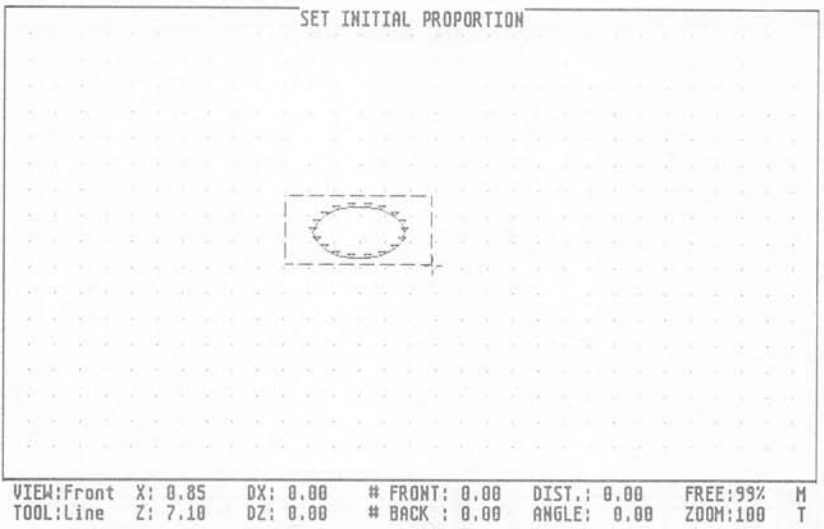
Fig. 136 ...and set the INITIAL PROPORTION



Move the cursor and observe the rectangle drawn with broken lines that grows from the fixed point.

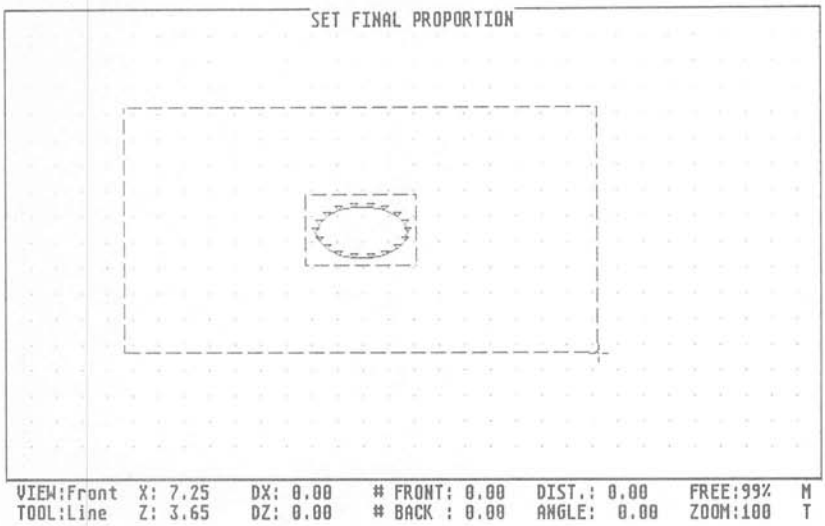
Define a rectangle that contains the selected figure and click the mouse button to confirm. Observe the message *SET FINAL PROPORTION*.

Fig. 137 Click the left mouse button after enclosing the object



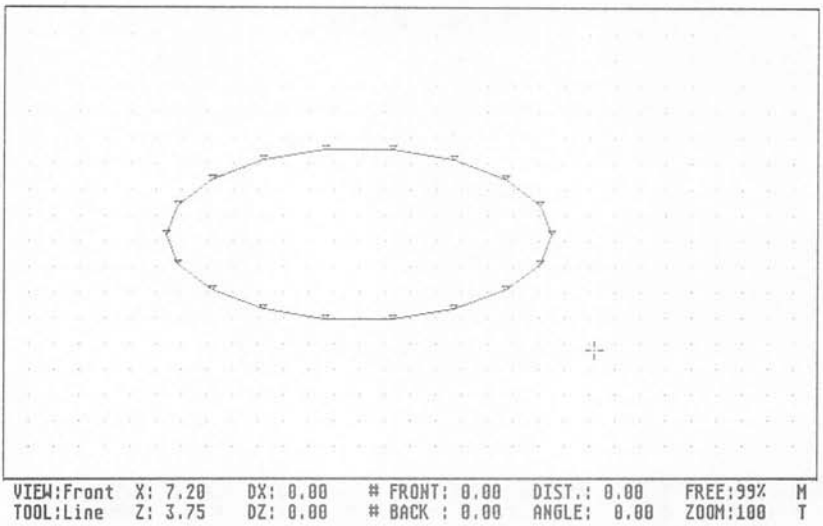
Move the cursor again and observe the broken-line rectangle that represents the new proportion (Fig. 138).

Fig. 138



Now define a rectangle larger than the first and click the mouse button to confirm (Fig. 139).

Fig. 139



Procedure : Copy n Objects

Open the CUBE.OBJ or execute the **CUBE** procedure found on page 18.

Open the **Select** menu and choose *Objects* (Fig. 140). Observe the message *SELECT OBJECTS AND PRESS RIGHT BUTTON TO CONTINUE.*

Fig. 140

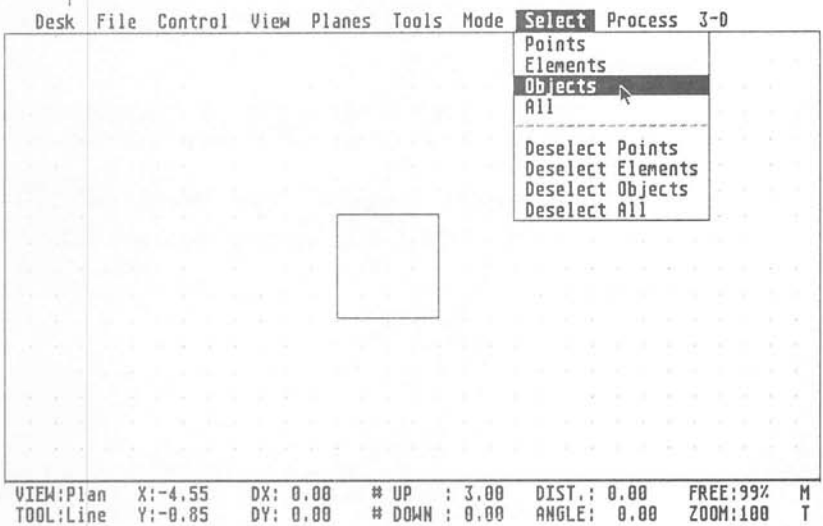
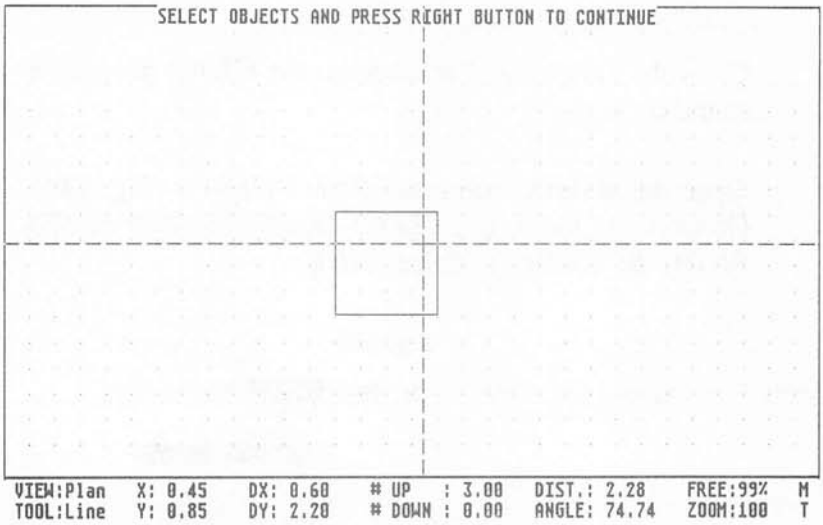


Fig. 141 Click the edge of the cube to select



Place the cursor on one of the edges of the object and click the mouse button to select it. Click the right mouse button to continue.

Open the **Process** menu, select *Copy n* (Fig. 142) and observe the message *SET INITIAL BASEPOINT POSITION* (Fig. 143).

Fig. 142

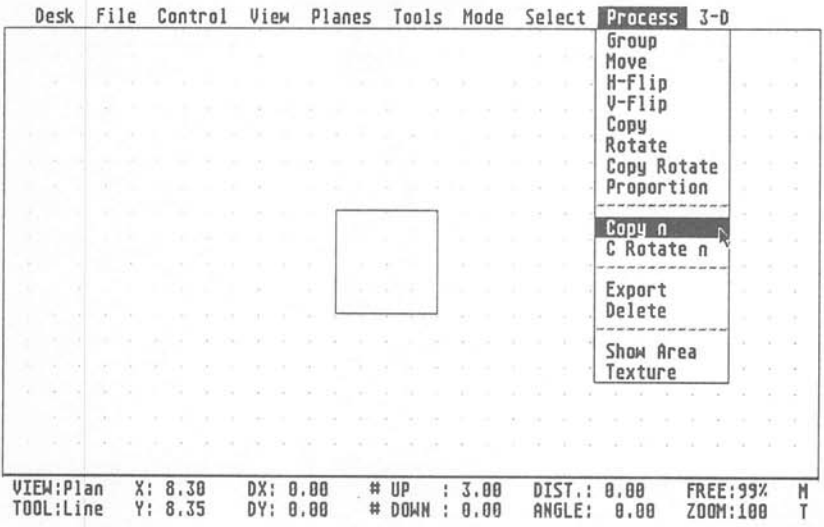
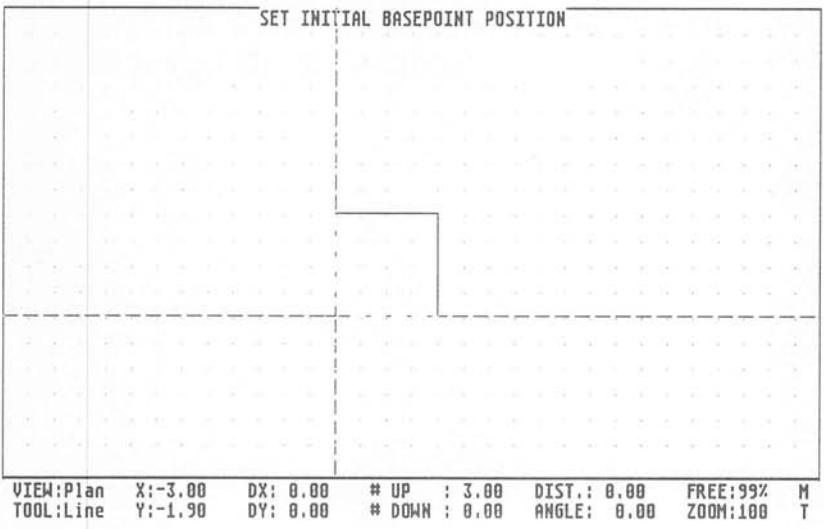
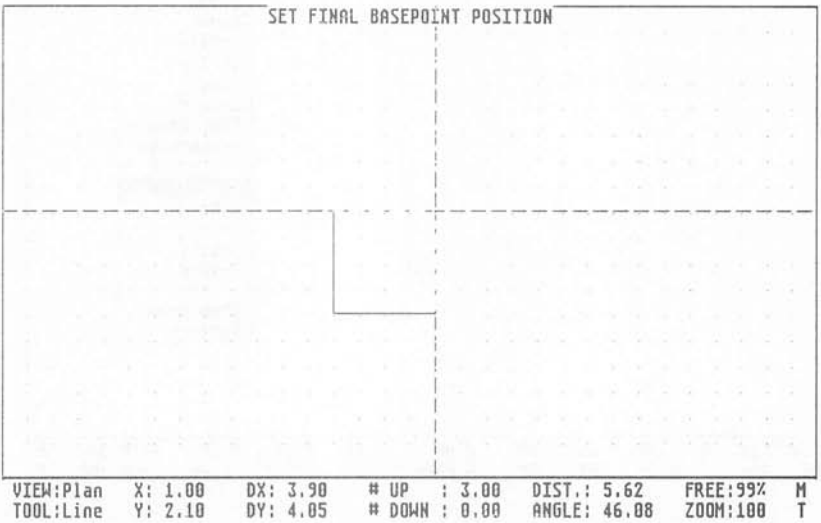


Fig. 143



Now place the cursor on the lower left corner of the object to take it as a point of reference, click the mouse to fix it and observe the message *SET FINAL BASEPOINT POSITION*.

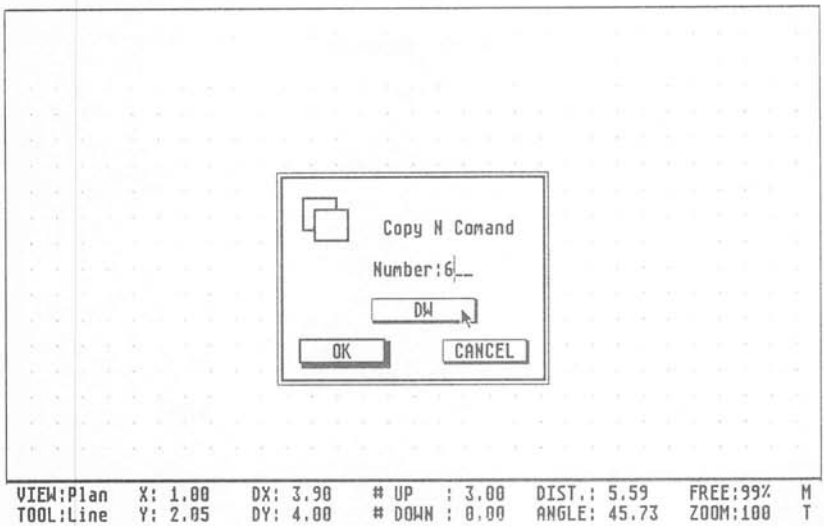
Fig. 144 Set the final Basepoint position



Place the cursor on the upper right corner of the object to mark it as a destination point of reference, click the mouse button to set it and you will see the dialog box *Copy n* COMMAND.

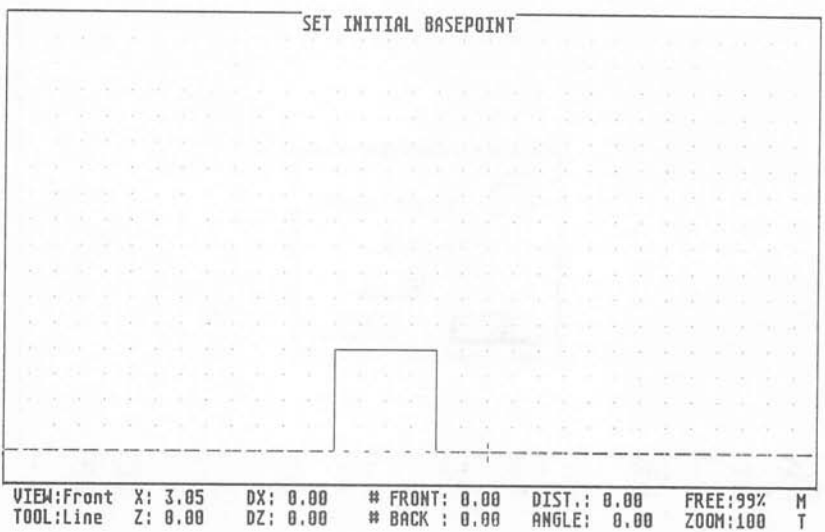
Press the <ESC> key and choose the number of copies that you wish (for the purposes of this tutorial, we recommend between three and eight copies).

Fig. 145 Enter Number of Copies (6)



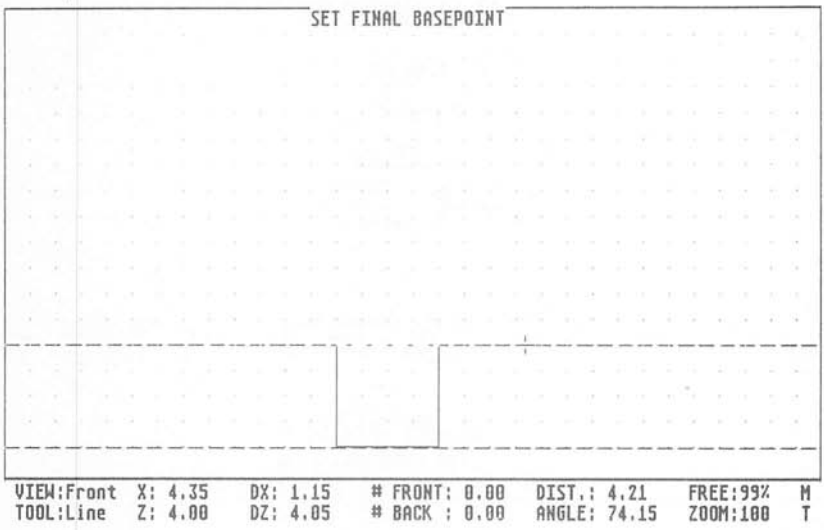
Now select the *DW* option to define the relative perpendicular position between the base and its immediate copy. Observe the message *SET INITIAL BASE POINT* and that the broken line that indicates the height of the cursor.

Fig. 146 Set the Initial Basepoint



Move the indicator until it touches the base of the object and click the mouse button to define the initial relative height. Observe the message *SET FINAL BASE POINT*.

Fig. 147 Set the Final Basepoint



Now place the indicator so that it touches the upper part of the object and click the mouse button to confirm.

The same dialog box will appear again permitting you to change the data if you wish. Select *OK* in this case to complete the operation.

Fig. 148 Select OK

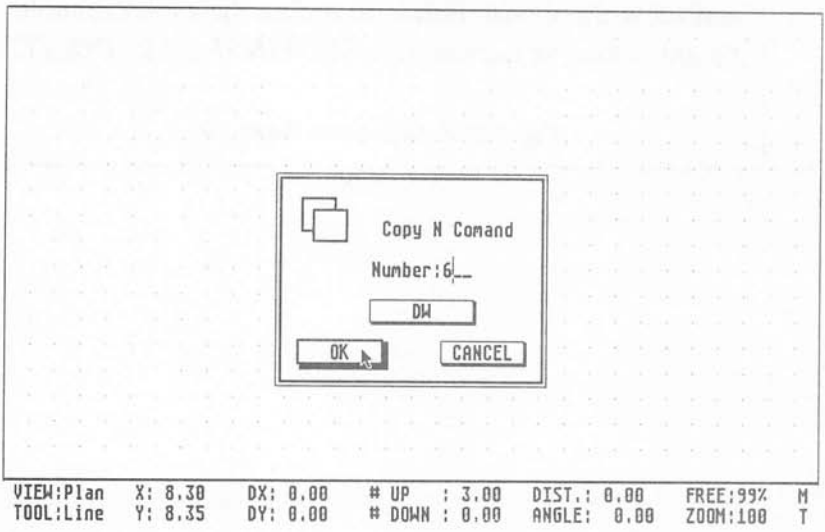
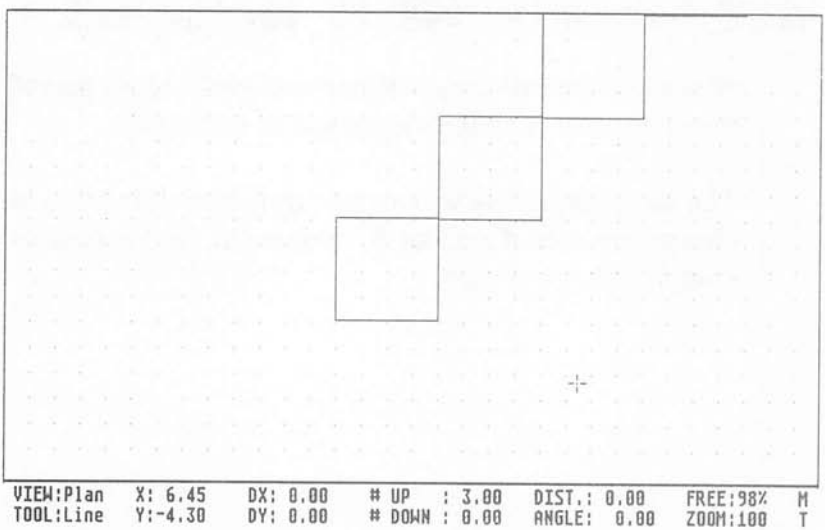


Fig. 149...And Observe the stacked cubes



Use the procedure **GENERAL VIEW** from page 24 to look at your new creation, then save it if you wish (***Warning:*** Make sure you save the new drawing under a different name as you will probably have numerous occasions for calling up the CUBE.OBJ file – *beginning with the next procedure*).

Fig. 150

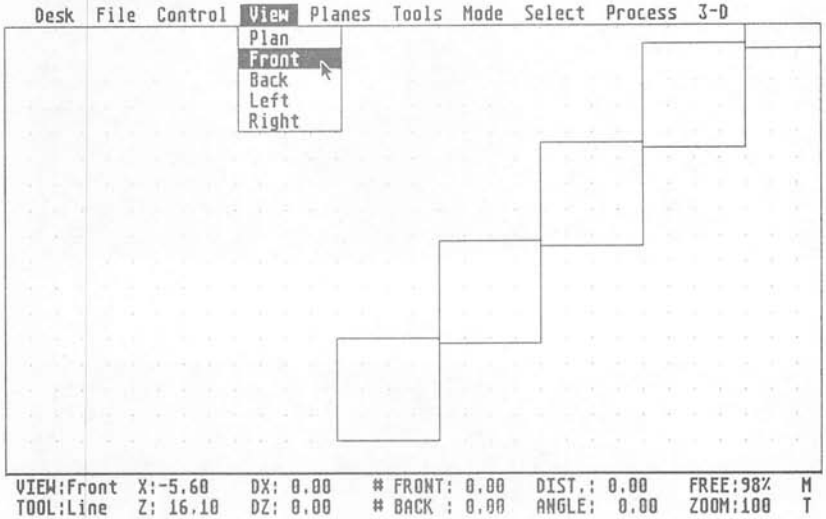


Fig. 151

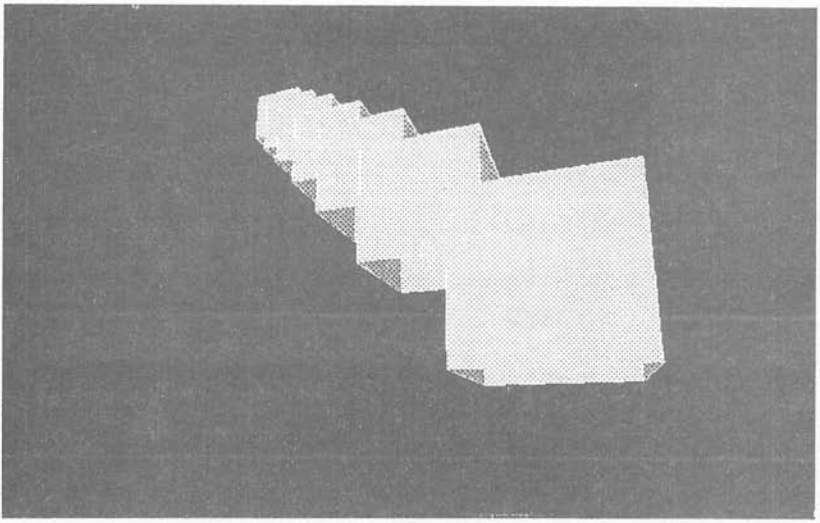


Fig. 152

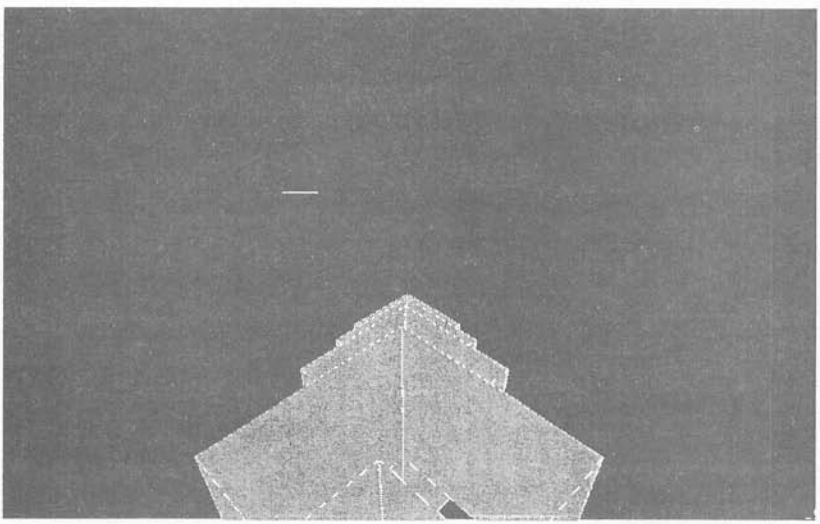


Fig. 153

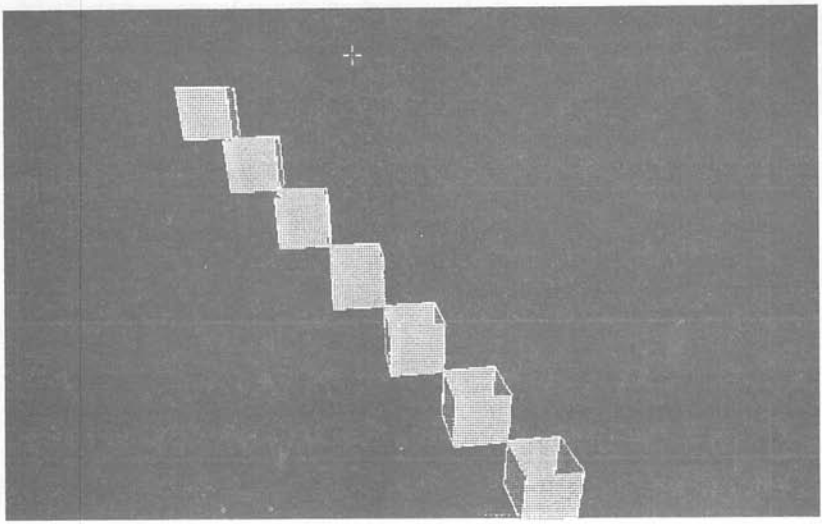


Fig. 154

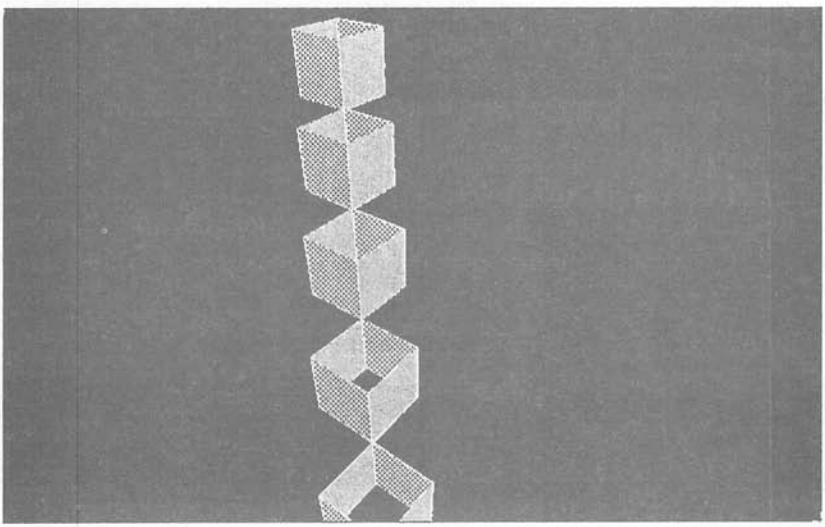


Fig. 155

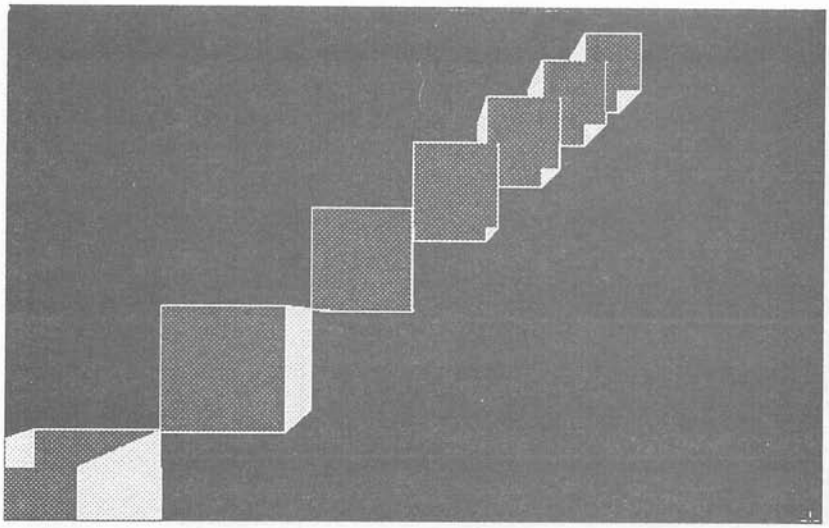
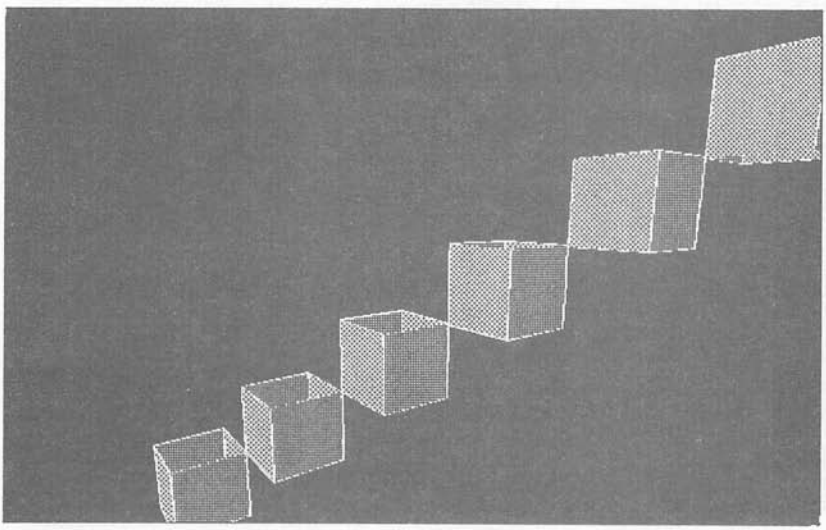


Fig. 156

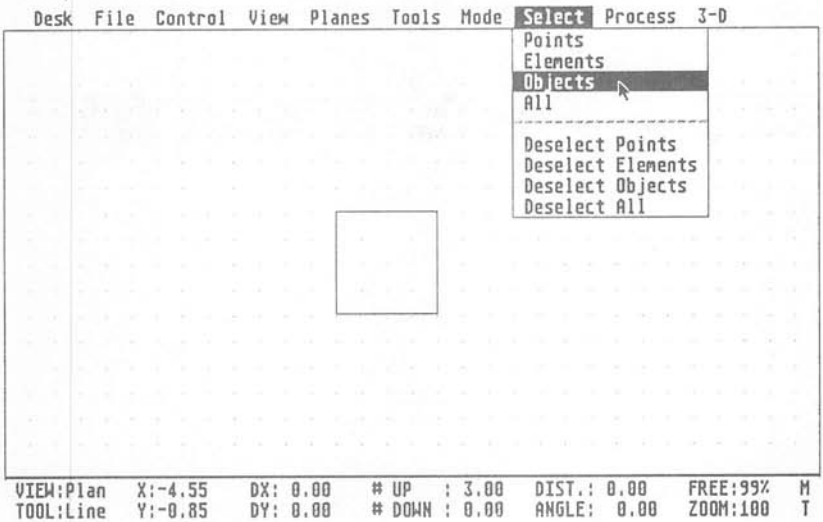


Procedure : Copy Rotate *n* Objects

Open the CUBE.OBJ again or execute the procedure CUBE without executing **GENERAL VIEW.**

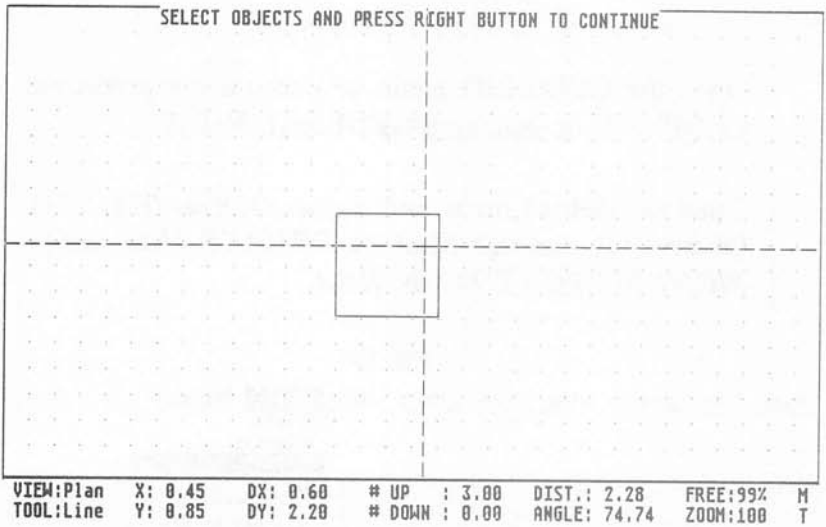
Open the **Select** menu and choose *Objects (*Fig. 157). Observe the message *SELECT OBJECTS AND PRESS RIGHT BUTTON TO CONTINUE.*

Fig. 157



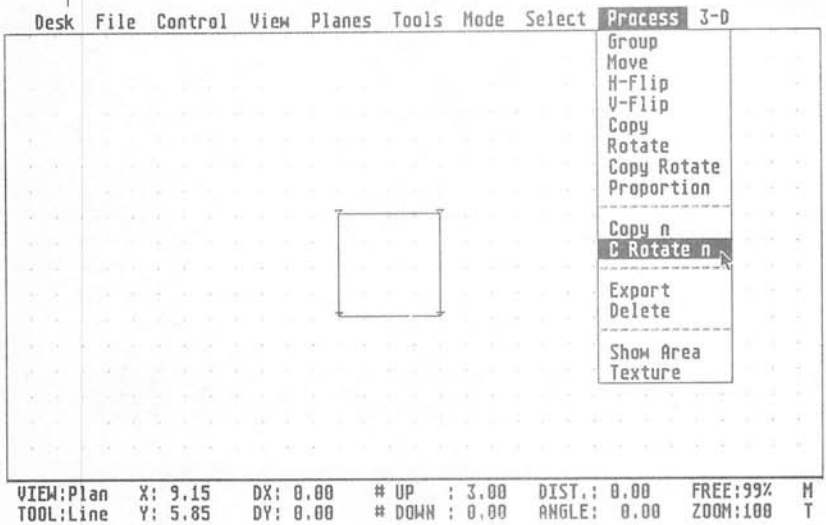
Place the cursor over one of the edges of the object and click the mouse button to select it (Fig. 158). Click the right button to continue.

Fig. 158



Open the **Process** menu, select C *Rotate n* (Fig. 159) and observe the message *SET SPIN CENTER*.

Fig. 159



Now place the cursor over the lower left corner of the object to mark it as a point of reference, click the mouse button to set it. ***MasterCAD*** shows you the angle selection window. Press the *Graphic Mode* button to continue and observe the message *SET INITIAL ANGULAR POSITION*.

Fig. 160 Set the Spin Center...

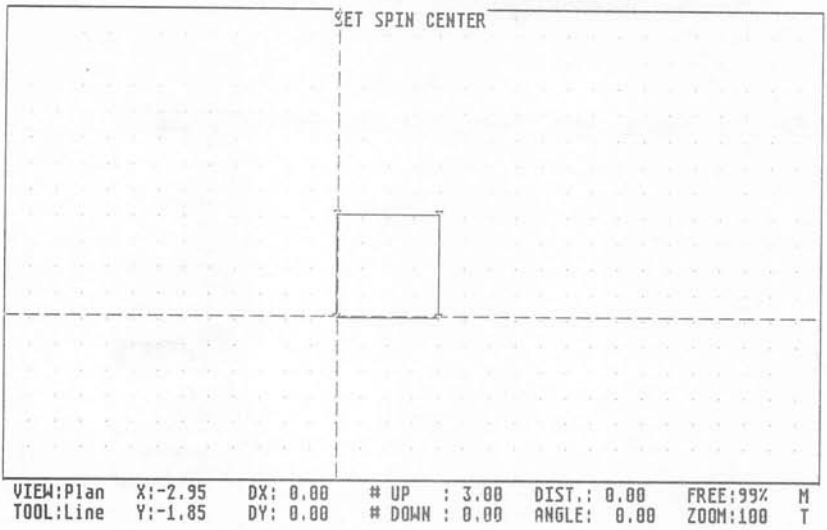
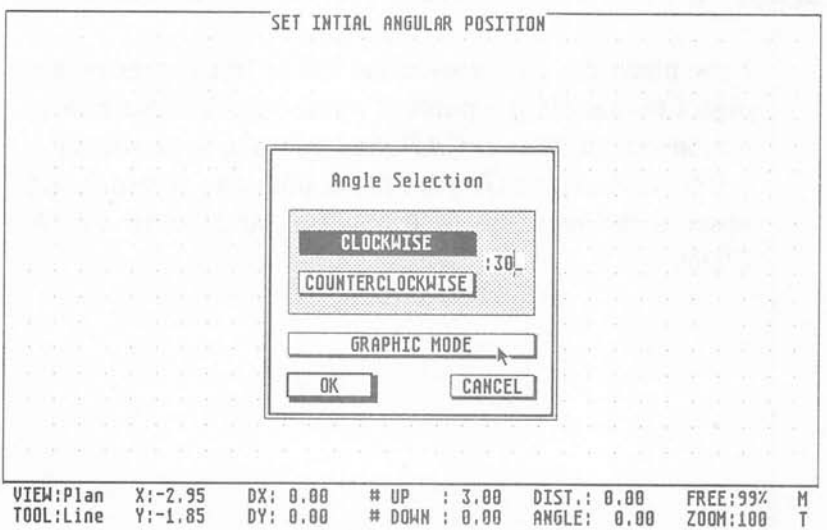
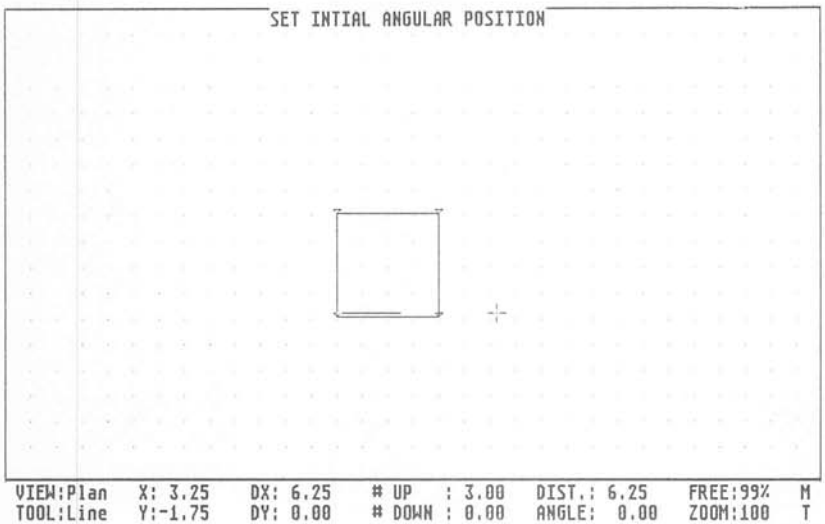


Fig. 161 ... Then select Graphic Mode



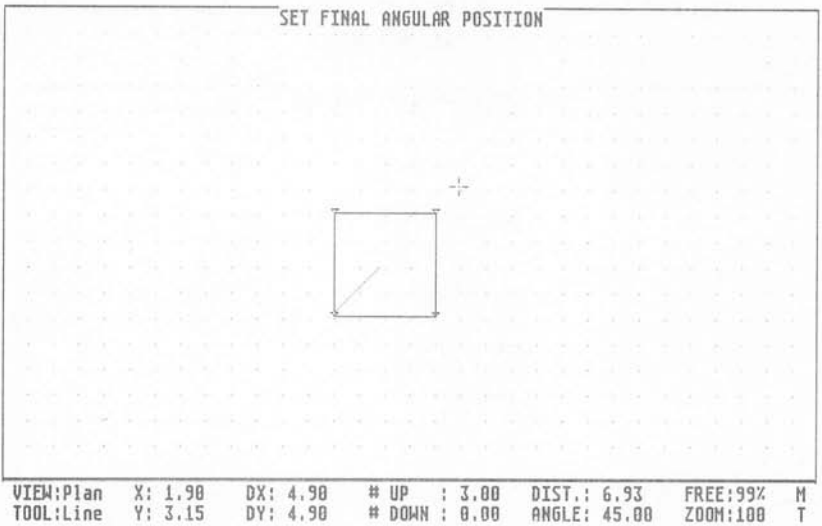
Move the rotation indicator so that it touches the base of the object (ANGLE =0) and click the mouse button to set it. Observe the message *SET FINAL ANGULAR POSITION*.

Fig. 162 Set the Initial Angle at 0.00 degrees (see the reading in the Monitor)



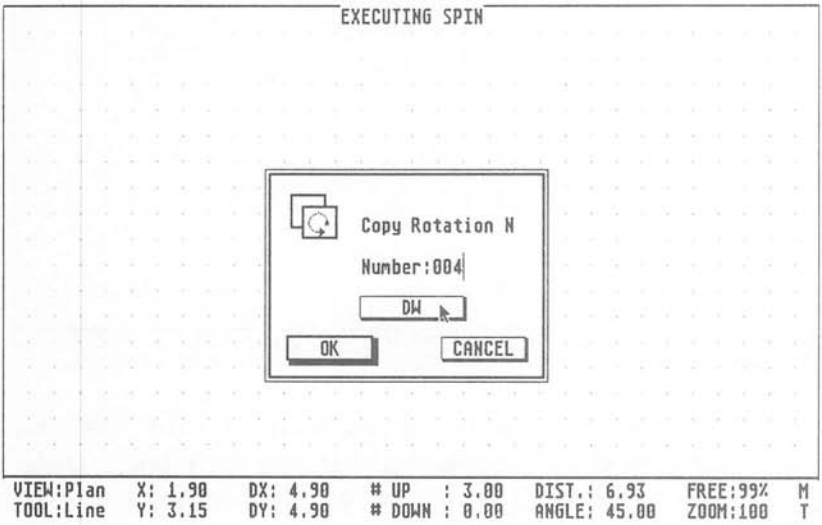
Move the rotation indicator 45 degrees (ANGLE = 45) and click the mouse button to set it. Carefully observe the dialog box *COPY ROTATION n*.

Fig. 163 Set the final angle at 45°



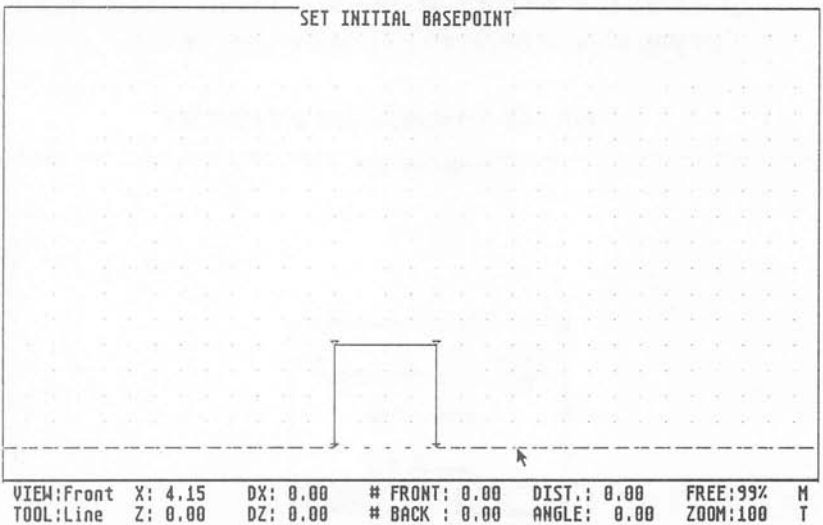
Press the <ESC> key and choose the number of copies that you wish (for the purposes of this tutorial we recommend that you choose between four and eight copies).

Fig. 164 Set number at 4 and click DW



Now select the *DW* option to define the relative height between the base object and its immediate copy. Observe the message *SET INITIAL BASE POINT* and the broken line that indicates the height of the cursor.

Fig. 165 Set the Initial Basepoint



Move the indicator until it touches the base of the object and click the mouse button to define the relative initial height. Observe the message *SET FINAL BASE POINT*.

Now place the indicator touching the upper part of the object and click the mouse button to confirm. Observe again the dialog box that permits you to change the data if you so wish. In this case, select *OK* to complete the operation.

Fig. 166 Set the Final Basepoint...

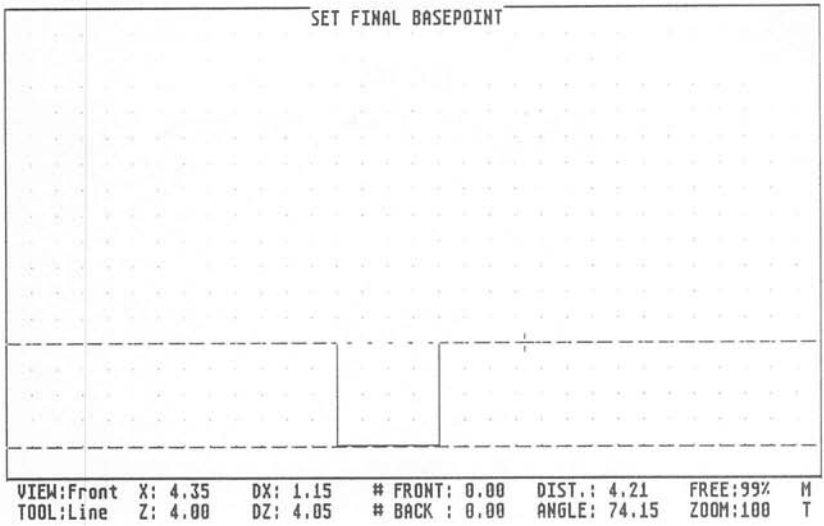
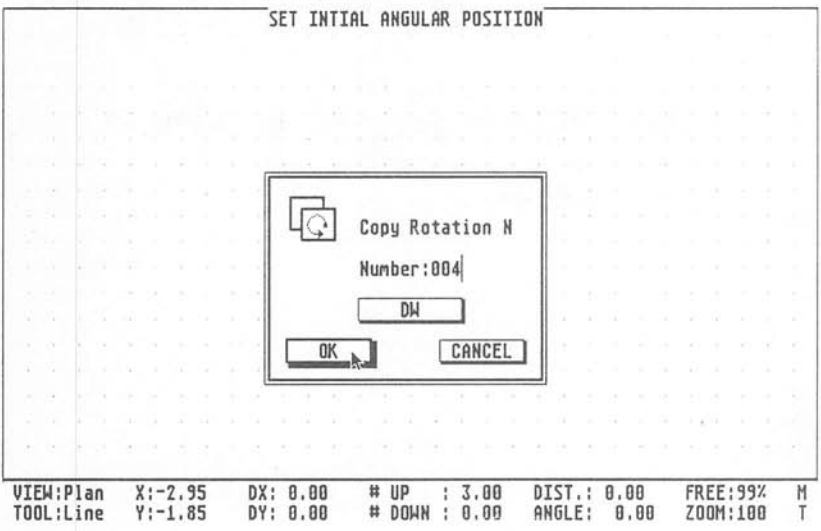


Fig. 167 ...And Click OK



Execute the **GENERAL VIEW** procedure to see the results.

Fig. 168

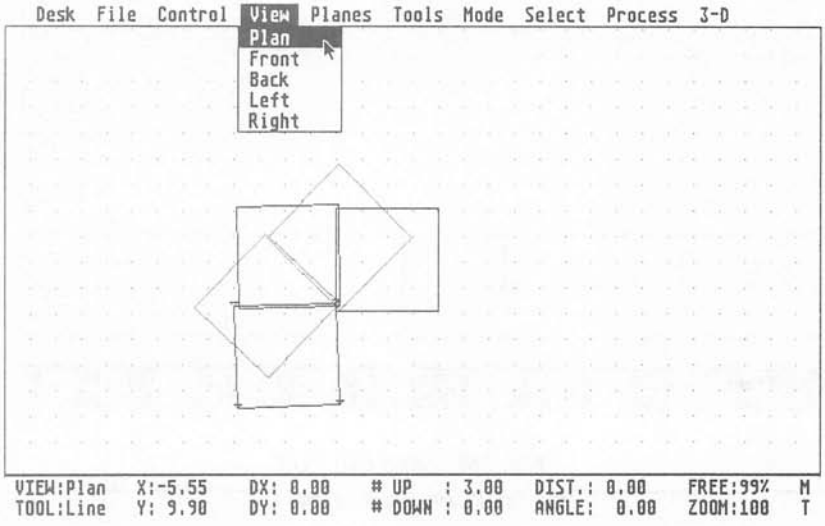


Fig. 169

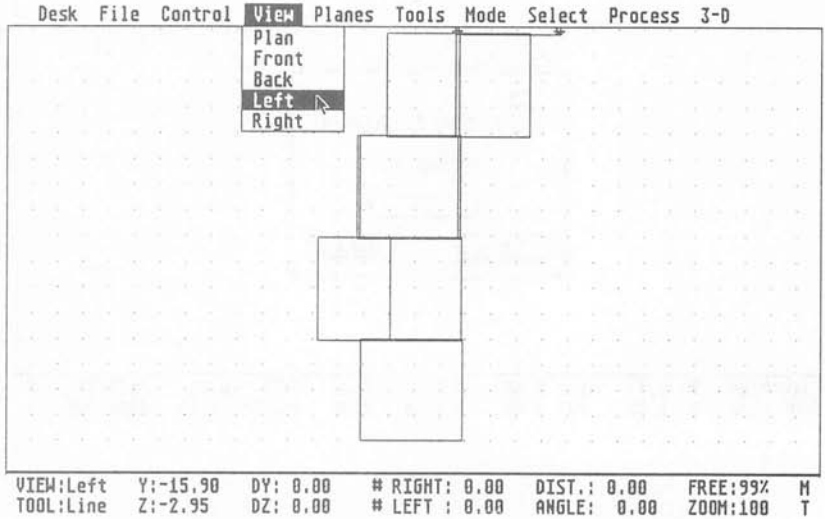
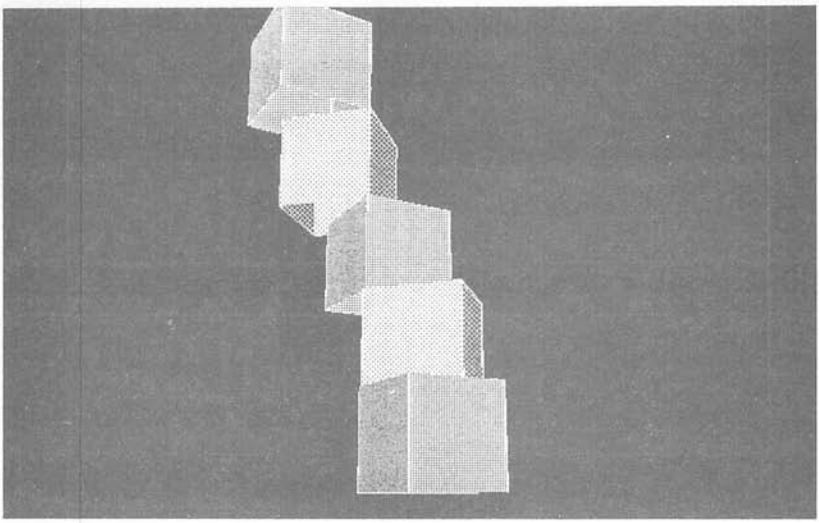


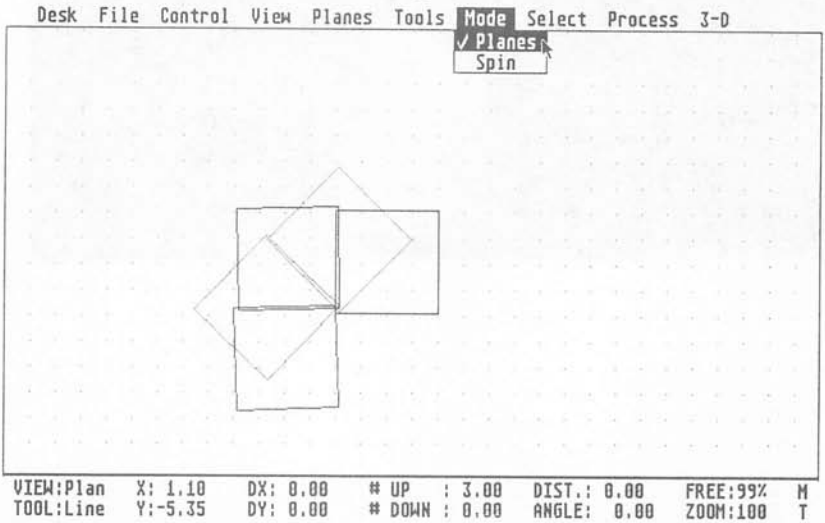
Fig. 170



Procedure : Mode

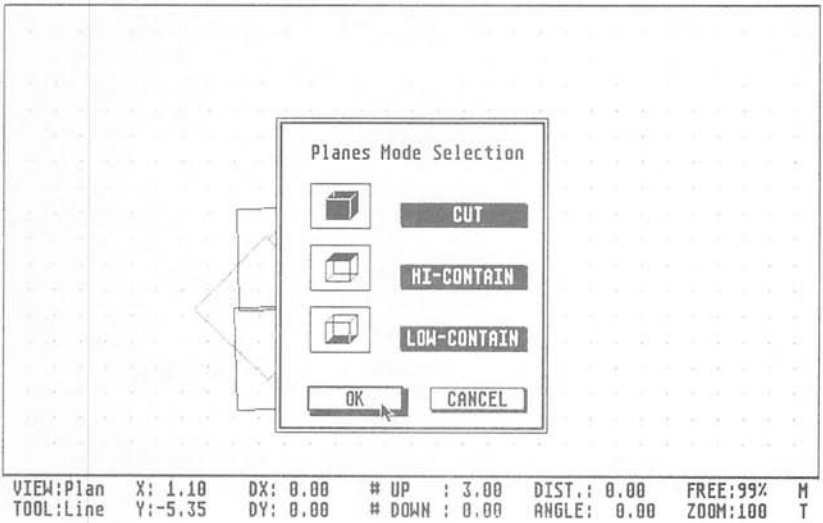
Open the **Mode** menu and select the *PLANES* option (Fig. 171). Carefully observe the window *PLANES MODE SELECTION*.

Fig. 171



Up to this point, you have been working in the *Cut* mode and have observed that the objects created are open at the sides. This is because the projection mode can be presented in several different ways. Taking CUBE as an example, you will notice that it appears to be a solid, open at the top and bottom (*Cut* mode); or that it appears with a solid base with no top (*Cut* + *Lo-Contain);* or that it appears as a completely solid object (*Cut* + *Lo-Contain* + Hi-*Contain).*

Fig. 172 The **Planes** *selection Box with all options selected*



Now select *Hi-Contain* and observe that the *Cut* option remains selected. This is because the mode functions are cumulative; that is, they can be combined. You can now repeat this lesson utilizing the following modes:

CUT + HI-CONTAIN

CUT + LO-CONTAIN

HI-CONTAIN + LO-CONTAIN

CUT + HI-CONTAIN + LO-CONTAIN

Observe the difference in the objects when presented as solids.

REMEMBER:

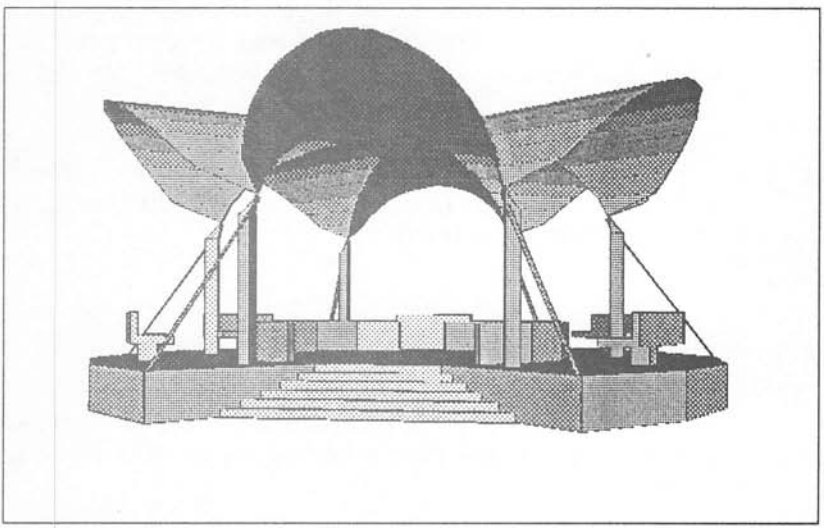
Lo-Contain and *Hi-Contain* are present when you are working on *Plan* view.

Front-Contain and *Back-Contain* are present when you are working on *Front* or *Back* views.

Left-Contain and *Right-Contain* are present when you are working on *Left* or *Right* views.

Chapter 3

Theory



CONCEPTS

Throughout this manual we use several terms like *projection, limit planes, perspective, projection lines, points, lines, rectangles, objects, elements,* etc.

This section will clear up their meanings (as much as possible), explain how to identify them, and, above all, show their interrelationships in the graphic environment we will call *workspace*.

Even if you are familiar with the terminology we will use from now on, it is always a good idea to go through it once more, further developing the bond between you and the program.

If you aren't familiar with this subject, read on carefully. These next few pages will introduce you to the fascinating world of graphic expression.

PROJECTION

Projection is the graphic representation of an object on a plane where all the points belonging to the object are projected.

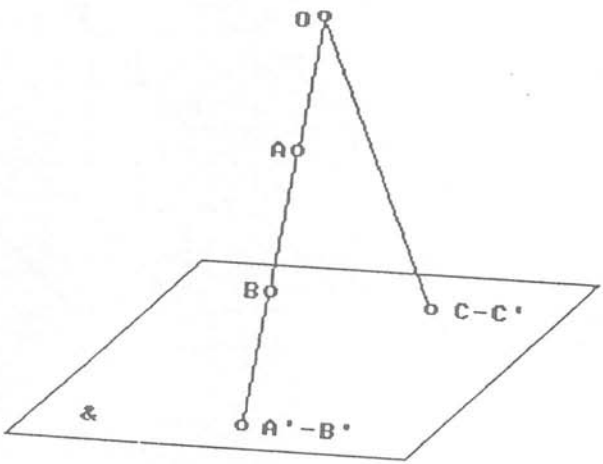
Professor F. Izquierdo Asensi, in his book Descriptive Geometry (11th edition, Chapter 1) defines *Projection* in the following terms:

> The projection of a point [A] in space (See Fig. 173) upon a plane [&], from a fixed point [O], outside the plane, is the intersection [A'] of the projection line [OA] (de-

fined by [O] and the given point [A]) with the plane [&]. The fixed point [O] is called the Projection Center and the plane [&], is called the Projection Plane.

From the above we can deduce that all points located on the projection line [OA], such as [B], will have the same projection [A'], and that if the point is located on the plane [&] it will merge with its projection, as we can see in [C]. This projection is called *Central Conic Projection or Perspective.*

Fig. 173



If the projection center is an "improper" point (meaning that it is located in infinity), the projection lines will be a parallel projection.

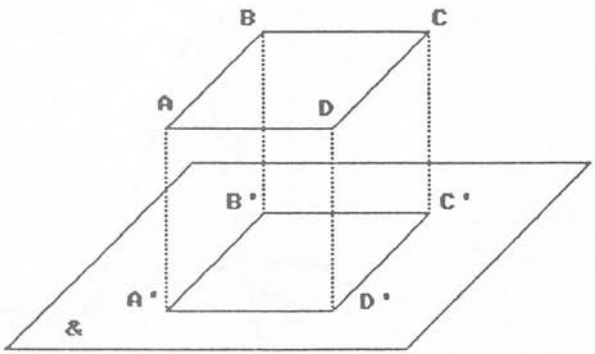
This brings us to the cylindrical or parallel projection. The cylindrical projection can be orthogonal or oblique, de-

pending on whether the common direction of the projection lines is perpendicular or oblique to the projection plane.

In the first case we have an ORTHOGONAL PROJECTION.

Fig. 174

ORTHOGONAL PROJECTION



PARALLEL PROJECTION

The projection lines are parallel and perpendicular to the projection plane, that we will now call the *picture plane.*

This type of projection is used to draw plane views, front and side sections, facades in architecture and engineering, and to project partial views of every type.

OBLIQUE PROJECTION

The projection lines are parallel and inclined to the projection plane.

This type of projection is used to draw technical and mechanical objects and is mainly used in engineering design. It is very useful because you can measure the object in 3-D.

PERSPECTIVE

Throughout this manual, when we refer to *perspective*, we are referring to a projection system called *conical linear perspective*. A practical derivative of the central projection, itself included in the conic system.

We shall then define *perspective* as a projection system by which an object in space is represented as it would appear in real life, with a three-dimensional appearance. It will be projected on a flat surface called the *picture plane* (see figure 175).

The effect of this projection can be compared to that of a photograph taken from any given point.

Applications

A perspective is used by an architect as an important component of the *Architect-Design-Aspect-Customer* relationship, since it affords a faithful rendition of the shape a building will have when it is finished. The technique is also useful in the fine arts, and in the production of advertising copy.

FEATURES

The outstanding difference between perspective and parallel projection is that the projection lines converge towards a common point called the viewpoint.

To achieve an adequate representation of the object, the projection system must use a series of reference elements that facilitate the building of images and their location in the workspace.

The elements that make up the perspective workspace are the following:

The Master CAD Coordinate System

This is a cartesian coordinate system made up by the three axes, X, Y and Z, that intersect at a common origin, located at the center of the screen in the plane X-Y (*plan view*) and in the lower center of the screen in the X-Z and Y-Z planes (*front, back* and *lateral views*).

The Viewpoint

This is the point in space from which the represented object is observed. The coordinate that establishes the position will be identified as XV, YV and ZV.

The View Angles

These are the vertical and horizontal angles that defines the observer's line of sight.

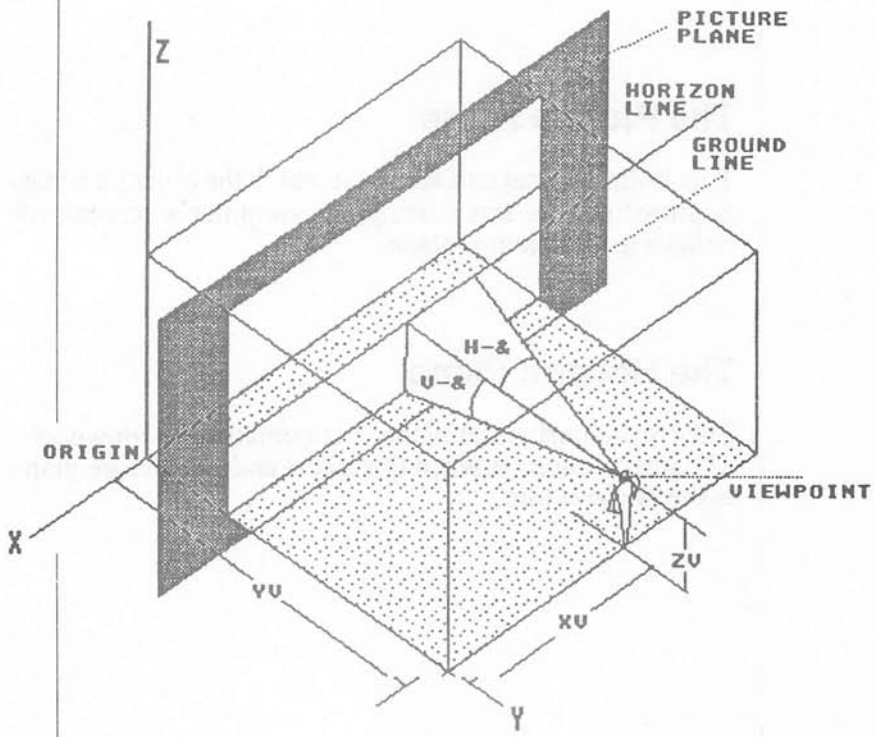
The Picture Plane

This is the vertical surface upon which the object's image is projected. In this case, your computer's screen will behave as the picture plane.

The Horizon Plane

This is the horizontal plane that contains the viewpoint. The intersection between this plane and the picture plane is the horizon line.

Fig. 175



WORKSPACE

H-& = Horizontal view angle V-& = Vertical view angle

The Ground Plane

This is a horizontal plane (floor), parallel to the horizon plane. Generally, this is the plane that contains all object bases and is the default ***Master CAD*** lower plane. It can be defined within the workspace as the set of points with a common Z coordinate equal to zero. The intersection of the ground and picture planes is the ground line.

The Vanishing Plane

This is the vertical plane that contains the viewpoint. It separates the spaces in front of and in back of the observer. For all practical projection effects, and thus for this program, you will be able to see only the space ahead of you; that is, all points that are in front of you and in the direction you have chosen in the **Viewpoint** menu.

The Ground Line

As we said before, this is the intersection of the horizon and picture planes. If the screen is taken as the ground plane, the ground line can be seen in *plan* as being in the picture plane's position.

The Horizon Line

This is the intersection of the horizon and picture planes. It is separated from the ground line by a distance ZV, corresponding to the height of the viewpoint. If we take the screen as the picture plane (see Figure 175, page 154),

we can see the ground line GL, the horizon line HL, the height of the viewpoint ZV, the fugue point FP, the distance points DP, and the distance DV, measured as the distance between the viewpoint and the picture plane.

The Central Fugue Point

This is the projection PC of the viewpoint on the picture plane. All lines parallel to the Y axis will converge to this point.

The Distance Points

These are located to the right and left of PC and at distance DV. All lines 45 degrees from the Y axis converge to these points.

The Limit Planes

These are user-defined planes that limit the projection of a given object in space. For example, if we have defined a horizontal plane with an elevation of three meters and a second horizontal plane at an elevation of seven meters and we draw (with ***Master CAD*** of course!) the plan of a circle, we will have an object with a height of four meters, elevated three meters above ground. These planes may be horizontal, vertical or angled as you wish.

If you are working in a *plan view*, you may then define horizontal or inclined limit planes that will determine the upper and lower limits of the object or objects you wish to project.

If you are working in a front or back view, you may then define horizontal or inclined limit planes that will determine the front and back limits of the object.

If you are working in a left or right view, you may then define horizontal or inclined limit planes that will determine the left and right limits of the object.

The limit plane is also used to draw graphics contained in them. In this case, a rectangle drawn on an inclined plane could represent, for example, the roof of a house; a cylinder drawn on a plane could be a disc in space, etc.

The Objects

These are sets of lines that make up any figure, and can be edited as a unit. They can be created in two different ways:

- from figures included in the **Tools** menu,
- combining several of the above using *Group Process*.

The Elements

These are straight lines or planes that make up objects. The former may be separated from the latter using the *Process* commands.

The Points

These are singular points that make up elements and can be selected and processed using the *Process* commands.

SPECIAL PROPERTIES OF THE PERSPECTIVE PROJECTION

The following are properties that define the ***Master CAD*** **Perspective Projection** and should be taken into account when using the program.

- A system of horizontal straight lines that converge at a single point is called the *Fugue Point* of the system.
- The horizon lines contain all the *Fugue Points.*
- The vertical lines (Z lines), being parallel to the picture plane, do not appear to converge (if the view angle is horizontal), and hence will be see as vertical in perspective.
- If the vertical view angle is inclined, then the vertical lines will converge to a single point, called the *Fugue Point of the Vertical System.*
- The lines contained in the picture plane will be actual size, since they will be their own perspective.
- In the same sense, if a line is located behind the picture plane, its perspective will be shorter than the line itself.

Some Tips

It is important to select an adequate viewpoint, since its location will determine the results of your perspective. A badly selected viewpoint can lead to unsatisfying results.

The location of the viewpoint depends on what part of the object you want to feature. It is a good idea to displace the object with respect to the viewpoint to avoid views that are too symmetrical.

When positioning an object with respect to the picture plane, it is preferable to place it in such a way that its side faces different angles within the same planes so that symmetrical views are avoided.

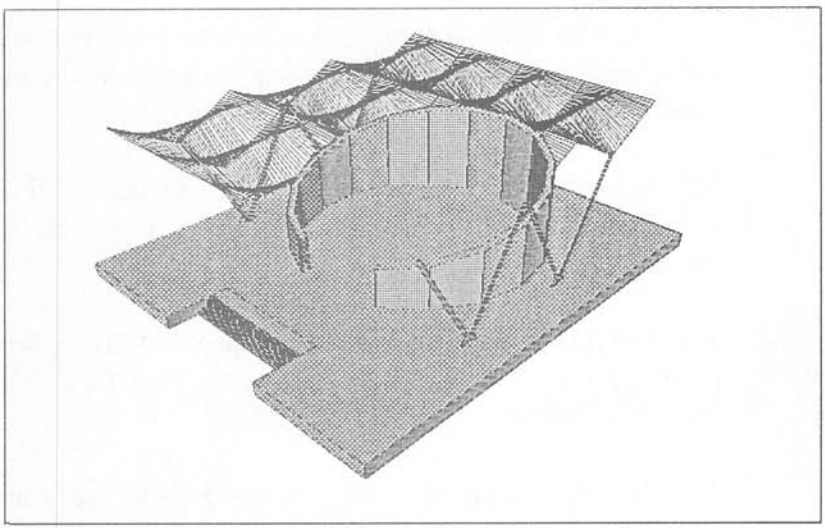
It is common to use angles of 30 or 60 degrees.

The height of the viewpoint above the ground plane is usually taken as five feet, nine inches (1.8m), that is, the height of the average person.

However, if the object to be viewed is small, use a height that is more convenient to you.

Chapter 4

Commands



Master CAD

Master CAD is a sophisticated *Computer Aided Design* system, conceived to let you and your *Atari ST* create complete and exact 2-D and 3-D graphics without having to waste endless hours building data matrices or fighting with mysterious and primitive user interfaces.

With ***Master CAD*** you will always know what you are doing because you will observe everything with your own eyes!

If there are any details of ***Master CAD*** that you cannot find in the manual, do not worry; ***Master CAD*** will provide on screen help at any time.

Master CAD is made up of two main programs designed to let you work in two work sections: the DESIGN SECTION and the OUTPUT SECTION.

The first is conceived to let you design and see your creation on the screen in an informal draft print; the second lets you obtain quality hard copy of your work on printers or plotters.

It is important to understand the capability of the drafting board. The *Atari ST* has three different screen resolutions: 640x400 monochrome, 640x200 RGB color, and 320x200 RGB color (***Master CAD*** does not run in the low-resolution, 320x200 mode).

Chapter 5: Commands

Master CAD uses the screen as a "window" into a larger workspace. The ***Master CAD*** workspace is a cartesian space with more than 4,000 kilometers in each positive or negative axis direction (that's about 2,500 miles!).

As you progress within ***Master CAD,*** you will create very sophisticated graphics more quickly each time. We strongly recommend that you save your work often to avoid disaster should your dog decides to munch unexpectedly on your power cord!

DESIGN SECTION

2-D MAIN MENU

FILE

Menu to select file-handling operations.

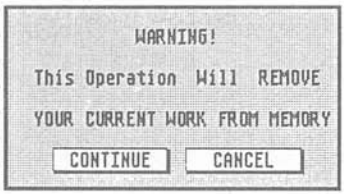
Fig. 176

NEW

The NEW command allows you to reset all parameters and default conditions. It also allows you to create a new workfile from scratch.

If you select the *New* file option and there is any workfile present in memory, you will see the following dialog box (w8).

Fig. 177



VIEW:Plan X:-13.65 DX: 0.00 # UP : 3.00 DIST.: 0.00 FREE:99% M
TOOL:Line Y:-2.60 DY: 0.00 # DOWN : 0.00 ANGLE: 0.00 ZOOM:100 T

You must decide what to do with the current file in memory.

If you CONTINUE the operation, you will lose all this information, so please be very careful when using this command.

If you decide to SAVE your work, ***Master CAD*** will go directly to the SAVE dialog box. The SAVE command will be discussed later.

If you CANCEL the operation the system will return to the drafting board, as if nothing had happened.

OPEN

This command opens a previously saved [.OBJ] workfile. It resets all work parameters and default conditions and erases any information contained in memory.

If you select this option while you still have a workfile in memory, you will see the same dialog box that you saw with the NEW command, so please be careful using this command.

Any file you open has to have been saved by ***Master CAD.*** After selecting the OPEN option, you will see a list of files which have the file extender [.OBJ]. You can open a file by typing the filename or by double clicking on the filename from the file directory.

APPEND

This command opens a previously saved [.OBJ] workfile and attachs it to the current file in memory. It doesn't reset the current parameters.

This option lets you merge two files together to create one larger file. When you append the second file to the first file, you want to make sure the combined size of both files doesn't exceed the memory limitation of your computer.

Check the amount of memory available and the size of the two files before beginning this operation, as ***Master CAD*** will only accept into the computer's memory that portion of the file that fits. The remainder will not load.

If you wish to *Append* a file, it has to have been saved using

Master CAD. After selecting the *Append* option, you will see a list of files which have the file extender [.OBJ].

You can append a file by typing the filename or by double clicking on the filename from the file directory.

Once the append operation is completed the appended file will appear selected, so you can place it wherever you want using the *Move* command of the **Process** menu.

IMPORT

This is a command to import objects from another file.

The objects being exported from the source file are saved in a buffer file named CONTAINER.OBJ that you can open if you want. If you wish to import the contents of the buffer file, it is necessary that the objects in that file were stored using ***Master CAD.***

Check first for the amount of available memory before using this command; ***Master CAD*** will accept only that part of the new file that fits in available memory.

To import the buffer file, open the **File** menu and select *Import*.

Once the import operation is completed, the contents of the buffer file will appear selected so you can place it where you want by using the *Move* command of the **Process** menu.

NOTE: If you attempt to import an empty Buffer file (before using the Export command) you will get a TOS error.

SAVE

This command saves to disk the object created as a FILENAME.OBJ.

If the file has been previously saved with a user-defined filename, this command will preserve that filename.

SAVE AS

Command to save to disk the file created with a user-defined filename.

When this option is selected, you will see the normal Item Selector with a list of files having the filename extender .OBJ.

If you are saving a file for the first time, you will have to enter a new filename. Be sure that the filename has the .OBJ extender; otherwise, the file will not be displayed the next time you select *Open*. When satisfied with the filename, select *OK* and the file will be saved.

DRAFT

This command sends a screen dump to the printer.

This option prints the picture in the current screen on your printer. You must have installed your printer driver properly (see the INSTALL PRINTER accessory on the desk menu).

If you press ALT+HELP during a draft print operation, the

printout will be canceled and the system will return to the workscreen. Please wait for the cursor and the monitor to appear before continuing to work.

SAVE SCREEN

This command saves the screen contents to different formats for quality print or plotting purposes. You can select two formats for saving the screen:

DEGAS format: The picture files saved with this format can be used by DEGAS" and other DEGAS compatible programs. The .i.DEGAS"; file is uncompressed and saved with the .PI3 extender for monochrome and .PI2 for RGB color.

MASTER CAD PRINTFILE: This is a special ***Master CAD*** file used by the OUTPUT section. If you want to print or plot your work, you need to save it in this format. These files are saved with a .PRF extender.

You have to define the zoom value in your printfile. This value defines the exact scale for plotting purposes. Depending on your printer, the ZOOM 1:89 may be printed in 1:100 scale; therefore, it is important to know the *aspect ratio* between the program and your printer. This must be tested and defined by the user.

When either of these options are selected, you will see the normal Item Selector with a list of files having the corresponding filename extender. If you are saving a new file for the first time, you will have to enter a new filename. When you are satisfied with the filename selected, select *OK* and that file will be saved.

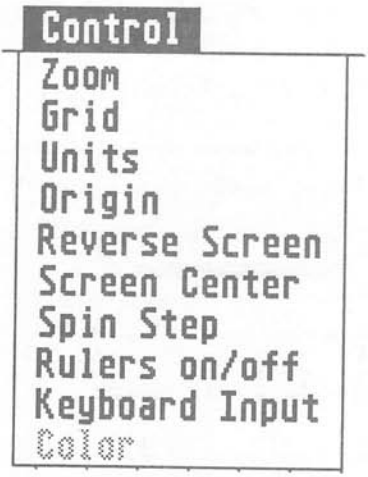
QUIT

Command to quit the DESIGN section and return to the Desktop.

CONTROL

Menu to define the work conditions and parameters.

Fig. 178



ZOOM

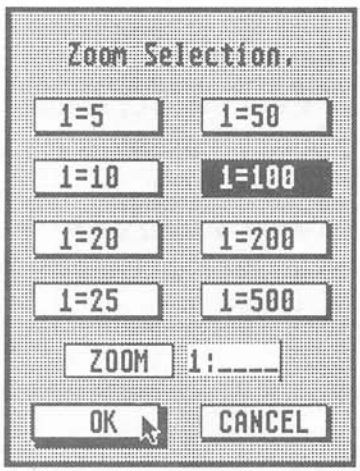
Command to modify the zoom. The range is from 1:1 to 1:1000.

We have named this command *Zoom* instead of *Scale* because we cannot guarantee the exact scale, given the endless list of printers on the market! You can be sure that the current zoom will give accurate plots on plotters supported by ***Master CAD.*** In the case of printers, you can adjust the zoom until you obtain the desired scale on your printouts.

To modify the zoom, open the Control menu and select the Zoom option. You will see a dialog box having different common zoom choices. Select the option you want and press OK to confirm.

If you want to define the zoom by input, press the *Zoom* option to activate it, enter the desired zoom number (from 1 to 1000), then press *OK* to confirm. If you enter a number greater than 1000, ***Master CAD*** will ignore it.

Fig. 179 ZOOM Select Box



Once the selection is made, you will see the objects on the screen in a new size (if the zoom selected is larger than before), or the new smaller screen that you can place by cursor movement (if you selected a smaller zoom than before).

You will see the message SET THE SCREEN CENTER BY CURSOR. Press the left mouse button to define the center.

Fig. 180 Set Screen


```
VIEW:Plan   X:-0.50    DX: 0.00    # UP   : 3.00   DIST.: 0.00    FREE:99%   M
TOOL:Line   Y: 10.60   DY: 0.00    # DOWN : 0.00   ANGLE:  0.00   ZOOM:200   T
```

GRID

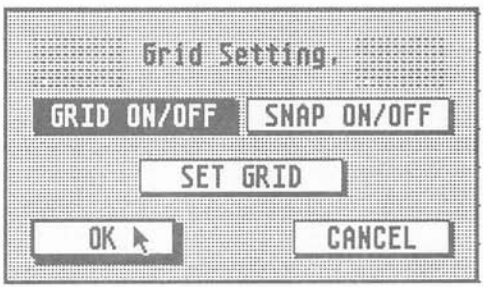
A grid of dots can be displayed on the screen for your reference. You will have three ways of using the grid:

GRID ON/OFF: Toggles the grid on and off.

SNAP ON/OFF: Toggles the snap function. The snap to grid function causes a user-defined point to be automatically aligned to the nearest grid point. This allows for greater accuracy in your graphics work. Once selected, the snap function is activated by pressing the SHIFT key while drawing.

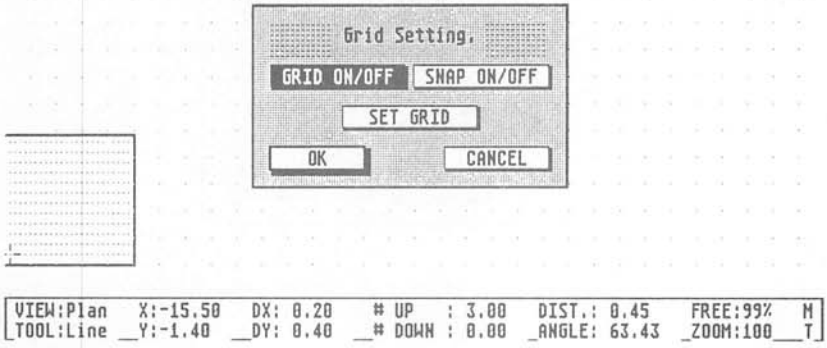
SET GRID: Sets the size of the grid. Select SET GRID from the dialog box, and you will see the message SET GRID and a square at the lower left corner of the screen.

Fig. 181 Grid Dialog Box



Move the cursor inside and to the lower left corner of this square. You can modify the grid display through cursor movement. You can also observe the grid size readings DX and DY or DZ to help you obtain greater accuracy. Once the grid has been defined, press the mouse button to verify your choice. The system will return automatically to the drafting board.

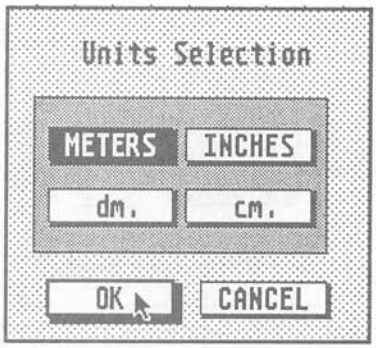
Fig. 182 Size Box



UNITS

This command lets your select the units of measurement you will use for objects displayed on the screen.

Fig. 183 Unit Selection

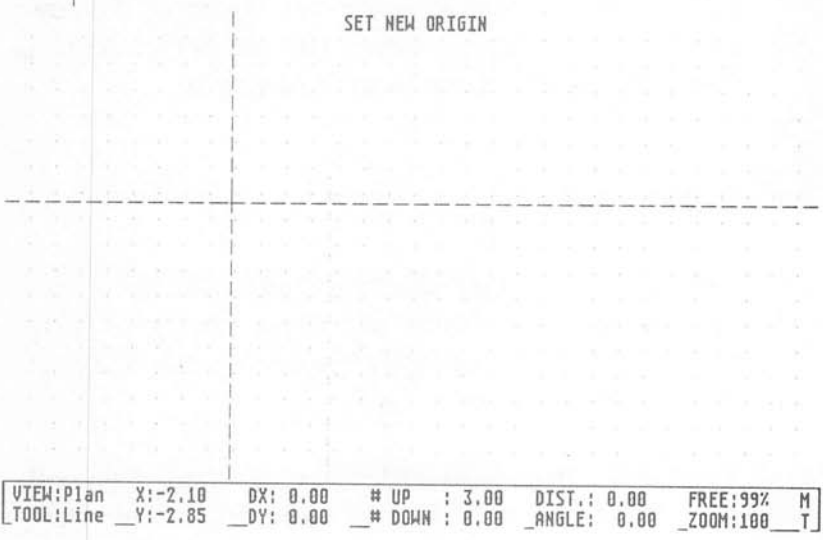


You have the following options: meters, centimeters, millimeters and inches. Select the option you prefer and *OK* to confirm your choice.

ORIGIN

Command to set a user-defined coordinate origin point. To define the new origin open the **Control** menu and select *Origin*. You will see the message *SET NEW ORIGIN*. Place the cursor where you wish the new origin to be and click the mouse button to confirm it.

Fig. 184



REVERSE

This command reverses the screen and its contents from white to black and vice versa (i.e., reverse video). Open the **Control** menu and select *Reverse Screen*. The screen will change instantly.

SCREEN CENTER

This command lets you move the window (screen) over the entire workspace. Open the **Control** menu and select *Screen Center*. You will see the message *SET NEW SCREEN CENTER*.

Move the cursor, and you will see a rectangle drawn with broken lines that represents the boundaries of the new screen. Place the screen center (the cursor) where you want and click the mouse button to define it.

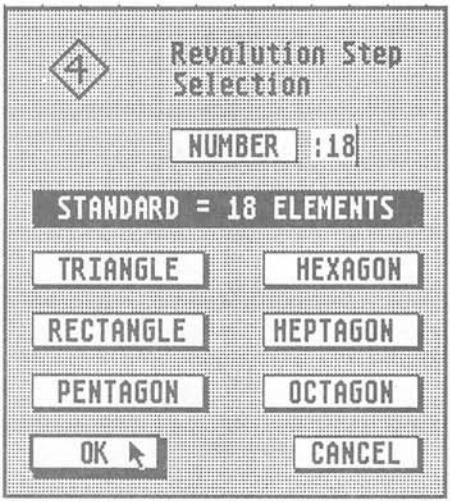
SPIN STEP

Command to select the regular polygon step number. This means the number of sides of a regular polygon. This value is used by the Regular Polygon tool and by the Spin object generation mode.

Open the **Control** menu and select *Spin Step*. You will see a dialog box that lets you select any of the more common regular polygons. Select the option you want and press OK to confirm.

You will observe that the default value for a circle is 18 sides, but you can modify this as you wish.

Fig. 185 Spin Step Dialog Box



If you want to define the *Spin Step* by input, select Number to activate it. Enter the desired spin step number (from 3 to 99) and press OK to confirm. If you enter a number greater than 99, ***Master CAD*** will ignore it.

RULER ON/OFF

This command toggles the graphic rulers. When the ruler is on, you can visually align figures by the markings on the ruler. You will also be able to adjust the spacing on the ruler by setting the grid (see SET GRID page 173). The ruler spacing will automatically match the grid spacing.

INPUT

This command selects the input mode. Open the control menu and select the input option, observe the dialog box indicating that you may choose between the graphic mode (default) and the keyboard mode. In the keyboard mode coordinates are entered directly from the keyboard.

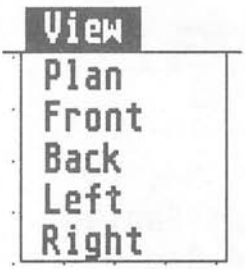
COLOR

This command selects the color to be used.

VIEW

Menu that selects the *orthogonal* view you will work with.

Fig. 186 View Menu



PLANES

Menu to determine the type of projection planes you will work with.

Fig. 187

NORMAL

Command to define projection planes that are parallel to the view plane that you are working with.

For example, when you define normal planes while working on the *Plan* view, these planes will be horizontal and obviously parallel to the floor.

If you define it while working on *Front* view, these planes will be vertical and parallel to the front projection plane.

To do this, open the **Planes** menu and select *Normal.* You will see the message *SET CUT PLANES a*nd a broken line that indicates the plane's level.

Move it until you are satisfied with the position of the plane, and then click the mouse button to confirm it. Repeat to position a second plane.

Note that if you are in *Plan* view, you are always going to define the normal planes in *Front* view because in *Plan* view you need only to set the height of the projection planes.

If you are working in *Front, Back, Left* or *Right* views you are always going to define the normal planes in the *Plan* view. That is because these planes will always be vertical.

PLAN ANGLED

Command to define projection planes that are vertical and inclined in respect to the view plane you are working with.

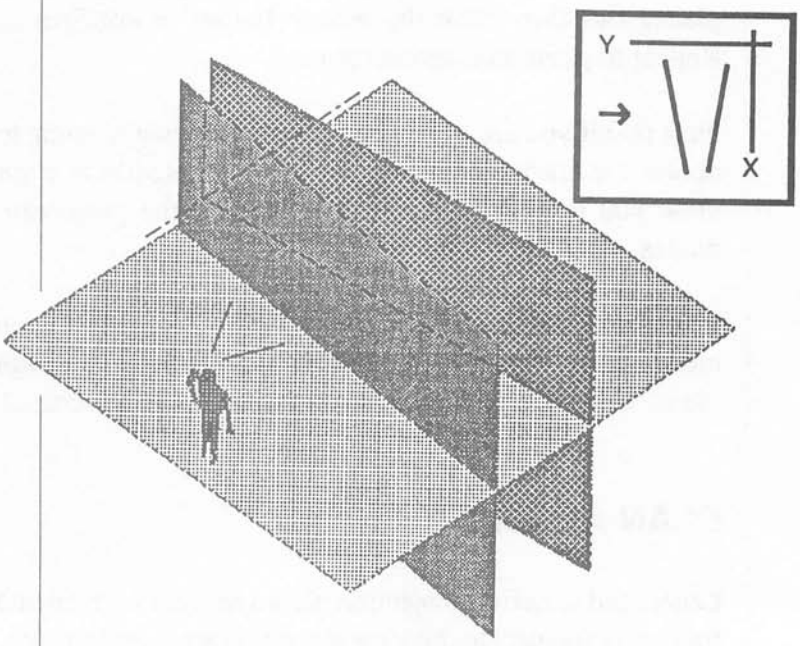
You can only define *Plan Angled* planes if you are working in *Front, Back, Left* or *Right* views.

To place them, open the **Planes** menu and select *Plan Angled.* You will see the message *SET CUT PLANES* and a broken line indicating the plane's position. Observe that you are in *Plan* view.

The *Plan Angled* planes have to be defined by setting two points of the line that represents the plan on the screen. These points also determine the angle between the cut plane and the view projection plane.

Repeat this operation for the second plane and ***Master CAD*** will automatically return to the view you are working on.

Fig. 188



PLAN ANGLED PLANES

FRONT ANGLED

Command to define projection planes that are inclined in respect to the horizontal plane.

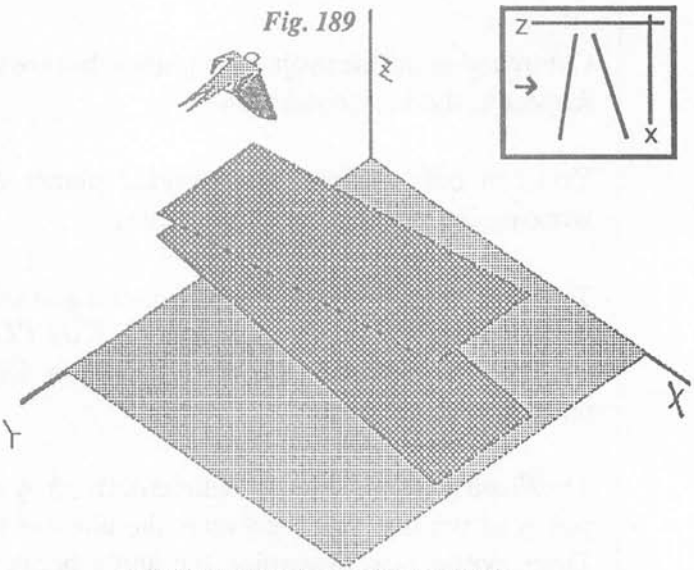
You can only define *Front Angled* planes if you are working in *Plan, Left* or *Right* views.

To place them, open the **Planes** menu and select *Front Angled.* You will see the message *SET CUT PLANES a*nd a broken line indicating the plane's position. Observe that you are in *Front* view.

The *Front Angled* planes have to be defined by setting two points of the line that represents the plan on the screen. These points also determine the angle between the cut plane and the view projection plane.

Repeat this operation for the second plane and ***Master CAD*** will automatically return to the view you are working on.

Fig. 189

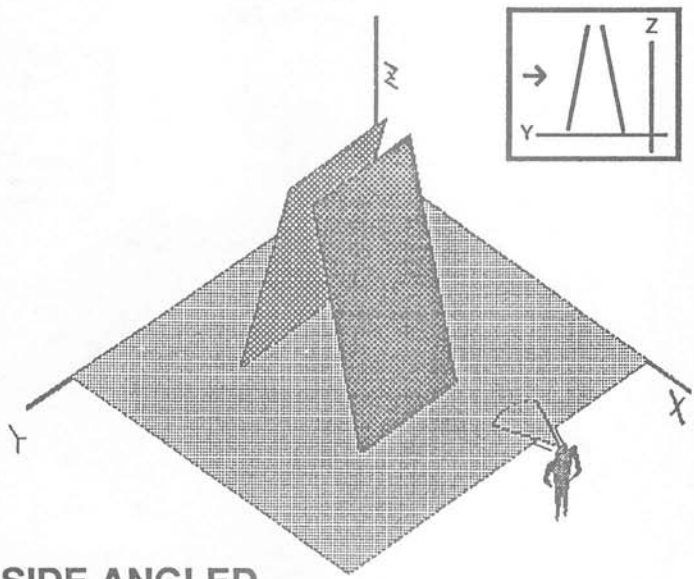


FRONT ANGLED planes viewed in PLAN

You can define *Front Angled* planes and use them from the *Right* view.

Front Angled planes seen from *Right* view.

Fig. 190



SIDE ANGLED

Command to define projection planes that are inclined in respect to the horizontal plane.

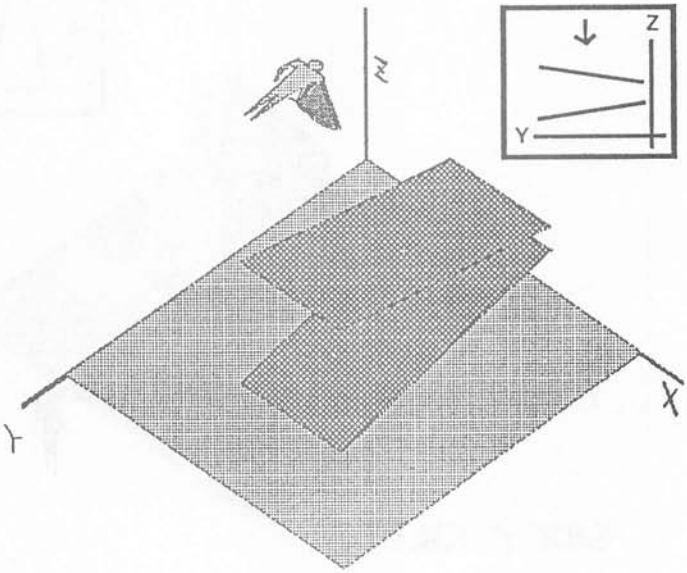
You can only define *Side Angled* planes if you are working in Plan, Front or Back views.

To place them, open the **Planes** menu and select *Side Angled*. You will see the message *SET CUT PLANES* and a broken line indicating the plane's position. Observe that you are in *Left* view.

The *Side Angled* planes have to be defined by setting two points of the line that represents the plan on the screen. These points also determine the angle between the cut plane and the view projection plane.

Repeat this operation for the second plane and ***Master CAD*** will automatically return to the view you are working on.

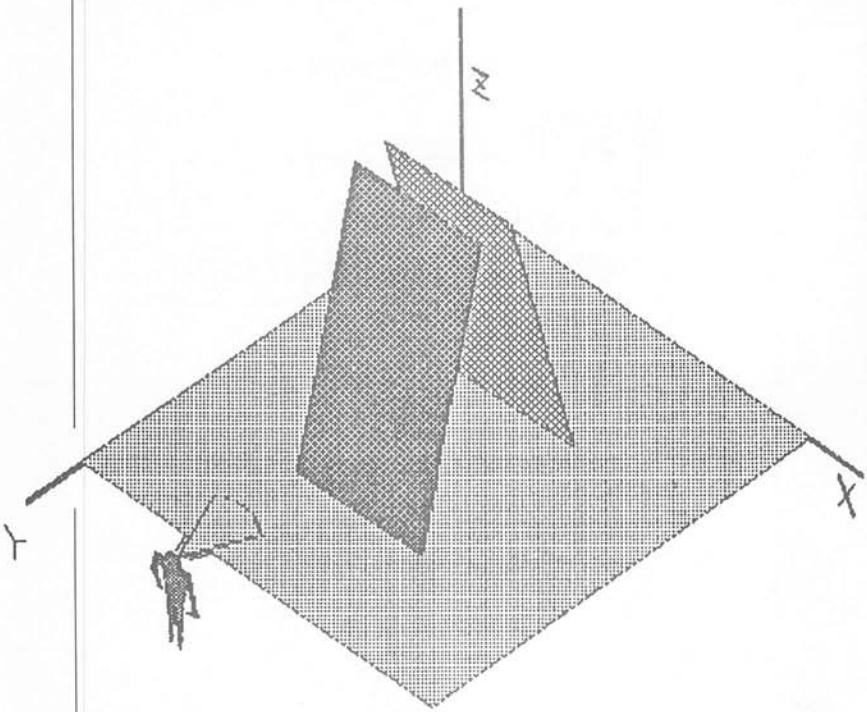
Fig. 191



You can define *Side Angled* planes and use them from *Front* view.

Side Angled planes seen from *Front* view.

Fig. 192

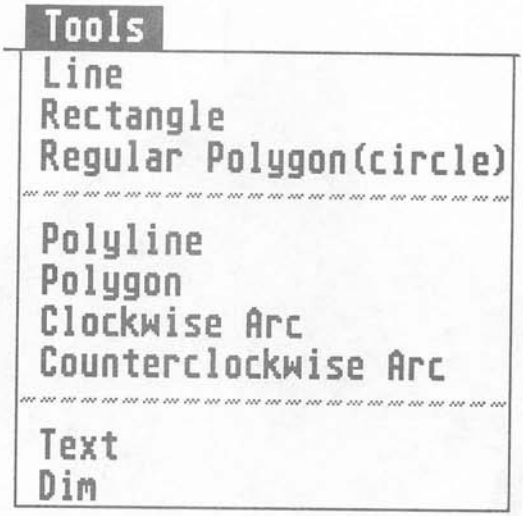


SIDE ANGLED planes seen from FRONT view

TOOLS

This Menu selects the graphic tool for use in drafting.

Fig. 193 Tools Menu



LINE

Command to define a straight line in any direction.

To draw a line, open the **Tools** menu and select the *Line* option. Position the cursor where you want the line to begin, and click the mouse button to confirm.

Move the mouse and note that you are drawing a line from the point you selected. If you decide to change the original point, press the right mouse button and start over.

To continue, place the cursor at the end point of the line and click the left mouse button to complete the line.

RECTANGLE

Command to draw rectangles.

Open the **Tools** menu and select *Rectangle*. Position the cursor where you want one corner of the box to be drawn. Now click the mouse button and move the mouse. Note that the rectangle gets larger. If you need to change the position of the first corner, click the right mouse button and start over.

Position the cursor where you want the opposite corner of the rectangle to be and click the left mouse button to complete the operation.

REGULAR POLYGON

Command to draw circles, ovals and regular polygons.

To draw a regular polygon open the **Tools** menu and select *Regular Polygon*. Position the cursor where you want the circle center to be and click the mouse button to set it. Then move the cursor and note an ellipse developing on the screen.

If you decide to change the center, press the right mouse button and start over.

Observe the readings DX, DY and DZ indicating the radiuses of the ellipse. When both radiuses are equal, press the left mouse button to complete the circle.

To draw an oval polygon, position the cursor where you want the center to be and click the left mouse button to set it. Move the cursor and note the oval developing on the screen.

Observe the readings DX, DY and DZ indicating the radius of the oval. When you are satisfied, press the left mouse button to complete the oval.

If you wish to draw a regular polygon or change the number of sides of the polygon, open the **Control** menu and select *Spin Step*.

Select the regular polygon option you want or select *Number* and enter the desired number. Press *OK* when ready to confirm.

POLYLINES

Command to draw continuous lines.

Polylines work exactly like regular lines but, unlike regular lines, continue drawing from the last point you pressed the left mouse button.

To finish the polyline, press the right mouse button.

POLYGON

Command to draw irregular shapes.

*POLYGON w*orks the same way as *Polylines,* with the exception that when you press the right mouse button, the last point drawn will be connected with the starting point.

CLOCKWISE ARC and

COUNTERCLOCKWISE ARC

Commands to draw clockwise and counterclockwise arcs.

Open the **Tools** menu and select *Clockwise Arc*. Position the cursor where you want to define the first point of the arc, then move the cursor and note that you can set the center of the arc through movement of the cursor. Position the center where you wish and press the left mouse button to confirm.

See the dialog box with the angle selecting options. You can enter the desired angle value and press *OK* to confirm or select the *Graphic Mode a*nd click *on* to continue.

In the second case, observe the radial indicator rotating around the center and the arc attached to the first point.. Set the second point as desired and press the left mouse button to complete the operation. Note that you are now in the polyline tool mode.

If you want to continue with arcs or any other poly tool, open the **Tools** menu and select the new tool. Note that the first point of the new tool will be attached to the last point of the previous tool. If you want to stop, just press the right mouse button.

TEXT

Command to write three-dimensional text.

Master CAD text is made up of straight lines and works like any other ***Master CAD*** object. It can be printed, even plotted.

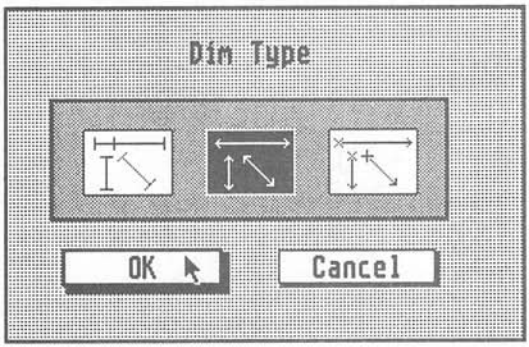
Open the **Tools** menu and select *Text*. Note that the monitor disappears, and the message *ENTER TEXT LINE AND PRESS RETURN is* displayed. Enter your text, press return and you will see the message *SET BASE POINT*.

Set the lower right corner of your text by moving the cursor and click the mouse button to confirm.

DIM

This command creates dimensioning references of the objects. Once you have selected *DIM* from the **Tools** menu, select the *DIM* type and parameters from the dialog box and press *OK* to confirm. The system will return to the work-table and will give a set of instructions according to the type of *DIM* tool selected.

Fig. 194 DIM Parameters Selection



MODE

Menu for selecting the drawing system:

Fig. 195 Mode Menu

PLANES

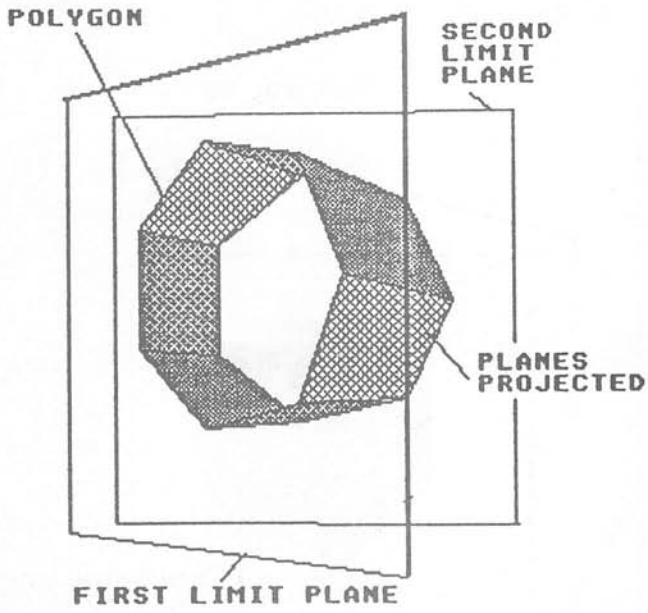
Menu to select the projection mode. The projection mode will define which polygons will be solid, and which transparent.

You can see the result of the operations made with the following commands when you use the *Filled Planes* viewmode of the 3-D section.

When you have defined your projection planes or limit planes you can use them in three different ways, alone or combined.

CUT: When you use the cut mode, the lines or polygons that you draw in 2-D will be projected between the two limit planes and only the planes projected from these lines or polygons will be solid.

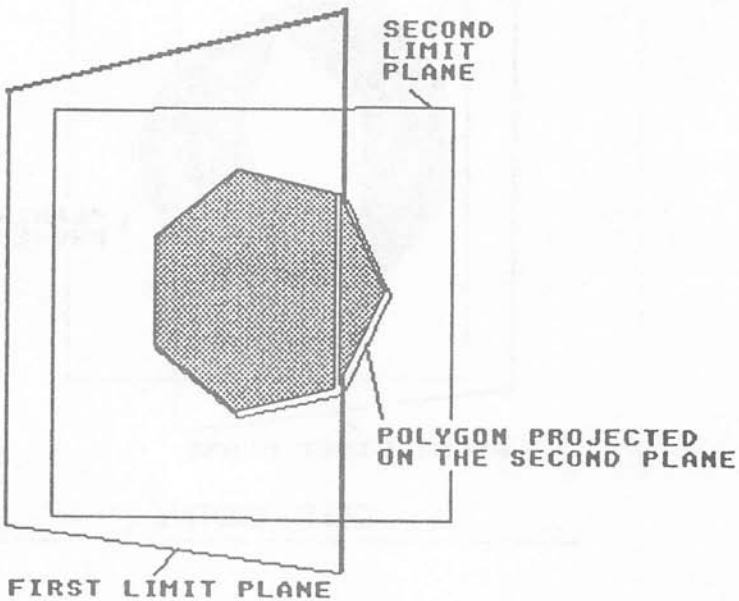
Fig. 196



LOW CONTAIN: When you use this mode, the lines and polygons drawn in 2-D will be projected only in the farthest defined planes.

If you make a polyline while working in low contain mode, it will function exactly like a polygon. If you wish to create isolated lines contained in any limit plane, use the *Line* tool.

Fig. 197

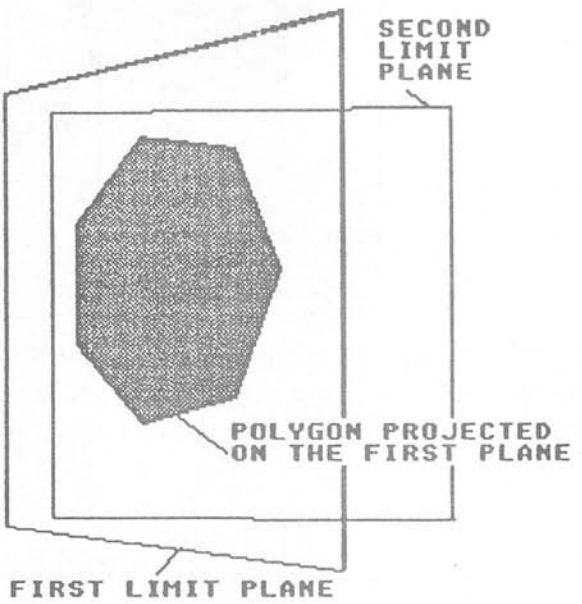


LOW-CONTAIN MODE

HIGH CONTAIN: When you use this mode, the lines and polygons drawn in 2-D will be projected only in the nearest limit planes you have defined.

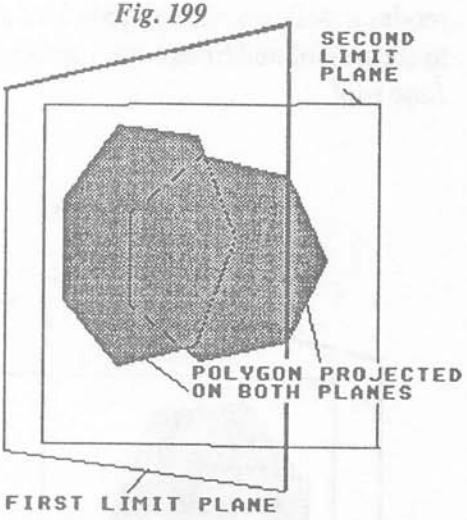
If you make a polyline while working in high contain mode, it will function exactly like a polygon. If you wish to create isolated lines contained in any limit plane, use the *Line* tool.

Fig. 198



HIGH CONTAIN + LOW CONTAIN: When you use this mode, the lines and polygons drawn in 2-D will be PROJECTED IN BOTH LIMIT PLANES YOU HAVE DEFINED, BUT THERE WILL NOT BE ANY PLANES PROJECTED BETWEEN THEM.

Fig. 199



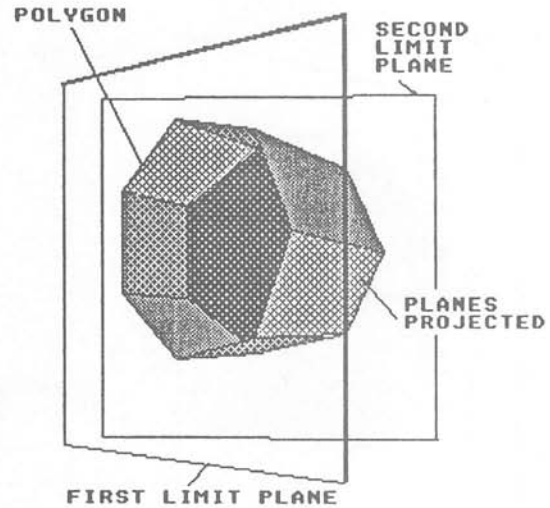
LOW+HI CONTAIN MODE

If you make a polyline while working in this mode, it will function exactly like a polygon. If you wish to create isolated lines contained in any limit plane, use the *Line* tool.

CUT+LOW CONTAIN: When you use this mode, the lines or polygons that you draw in 2-D will be projected between the two limit planes and will be projected in the farthest limit planes you have defined.

If you make a polyline while working in this mode, it will function exactly like a polygon. If you wish to create isolated lines contained in any limit plane, use the *Line* tool.

Fig. 200

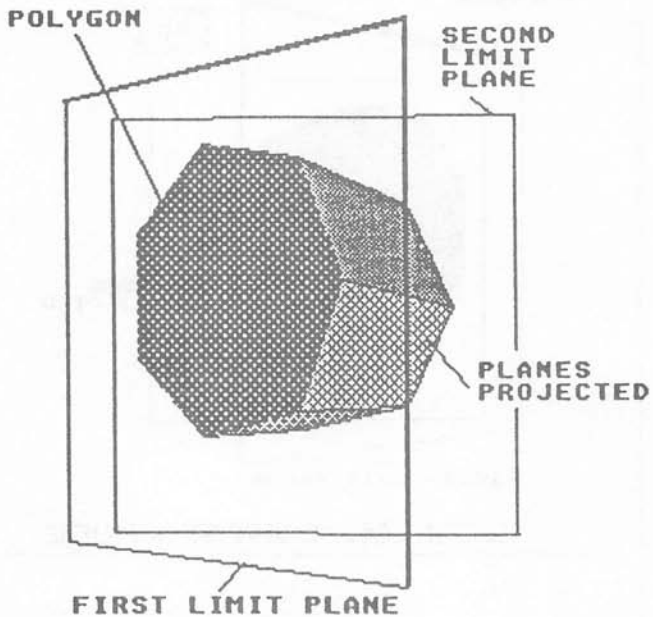


CUT+LOW CONTAIN MODE

CUT+HIGH CONTAIN: When you use this mode, the lines or polygons drawn in 2-D will be projected between the two limit planes and will be projected on the nearest limit plane you have defined.

If you make a polyline while working in this mode, it will function exactly like a polygon. If you wish to create isolated lines contained in any limit plane, use the *Line* tool.

Fig. 201



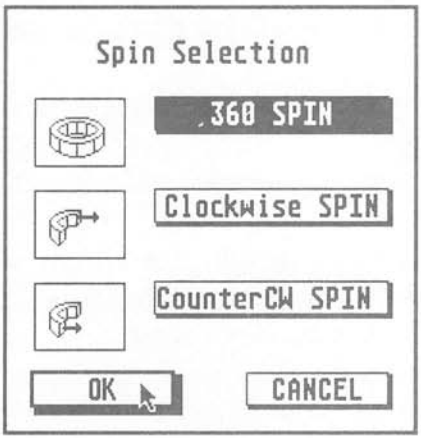
CUT + HI CONTAIN MODE

SPIN

Command to generate spun objects around a user-defined axis.

Open the **Mode** menu and select *Spin*.

Fig. 202 Spin Dialog Box



You have three spin types to choose from:

360 SPIN: Command generates a spun object 360 degrees around a user-defined axis.

Select *360 Spin* and then *OK* to confirm. You will see the message *SET SPIN CENTER WITH CURSOR*.

Position the cursor where you want the center to be, and click the mouse button to set it. Observe the vertical axis on the screen and the change in the VIEW reading. For example, if you set the spin center in *Plan* you will see the axis in *Front*.

You can now use any of the ***Master CAD*** graphic tools to make spun objects. For example, open the **Tools** menu and select *Rectangle*. Place the cursor where you wish to define the first corner of a rectangle and click the mouse button to confirm.

Now place the second corner and click the mouse again. You will see the message *CREATING SPUN OBJECT* and the spun object developing on the screen. Open the **View** menu and select the view you chose to define the spin center. You will see a circular shape.

If you want to change the number of sides of the spun object, open the **Control** menu, select *Spin Step,* enter the number of sides you wish, and select *OK* to confirm.

You can also set the spin center in any view. If you want to change it just open **Mode,** select the spin type and place the spin center where you wish.

By definition, a spun object (for select purposes) is created by the action of one tool around the spin center. For example, if you have drawn a line and a rectangle while creating spun objects, you will have two of them.

CLOCKWISE or COUNTERCLOCKWISE SPIN: Command to generate a spun object of *n* degrees around a user-defined axis.

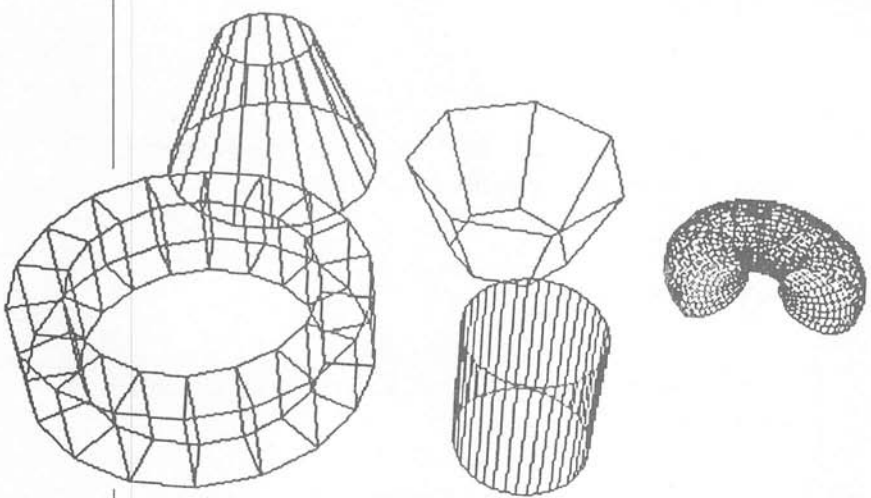
Select *Clockwise Spin,* for example, and then *OK* to confirm. You will see the message *SET SPIN CENTER BY CURSOR*. Position the cursor where you want the center

to be, and click the mouse button to set it. Note the rotating indicator around the spin center and the message *SET INITIAL ANGULAR POSITION*. Set the indicator where you want and click the mouse to confirm.

See the dialog box indicating the options for defining the arc's angle opening by numeric input or *Graphic Mode*. For the first option, simply enter the grade number and click on *OK* to confirm. In the second case, select the *Graphic Mode* and press *OK* to continue. Note again the rotational indicator stemming from the center point. Now you will see a new message, *SET THE FINAL ANGULAR POSITION*. Follow it using the cursor, and click the mouse button to complete the setting.

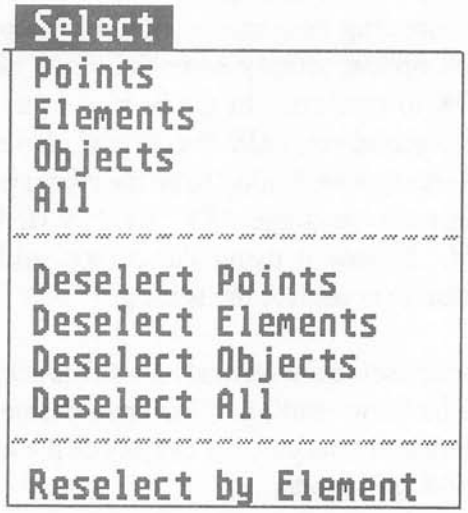
In both cases observe the vertical axis on the screen and the change in the *View* reading. You have defined a partial spin structure and now you can use any of the tools as you would in *360 Spin* mode.

Fig. 203 Examples of Spun Objects



SELECT

Fig. 204 **SELECT** *Menu*



Menu to choose the *Select* or *Deselect* criteria you will work with.

There are three types of "items" in the ***Master CAD*** system: *points, elements* and *objects.*

POINTS

These are singular points that make up elements and objects.

To select points, open the **Select** menu and then *Points.* You will see the message *SELECT POINT AND PRESS RIGHT MOUSE BUTTON TO CONTINUE,* and a cross-

type cursor on the screen.

Now you must define a rectangle that contains the points you want to select. Set the cursor where you want one corner of the rectangle to be and click the mouse button to confirm.

Move the cursor and note the broken-lined rectangle on the screen. Position the cursor to locate the opposite corner of the rectangle, and click the mouse button to complete the selection of the points contained in the rectangle.

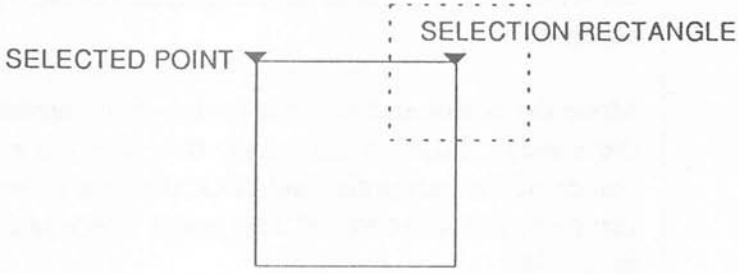
Note: Triangular shapes indicate the selected points.

You can repeat the selection process as many times as you wish. When satisfied, press the right mouse button to return to the drafting board.

To deselect points, open the **Select** menu and then *Deselect Points*. You will see the message *SELECT POINT AND PRESS THE RIGHT MOUSE BUTTON TO CONTINUE,* and a cross-type cursor on the screen.

The deselection process works exactly the same way as the selection operation.

Fig. 205



ELEMENTS

These are straight lines or planes that make up objects.

To select, open the **Select** menu and then *Elements*. You will see the message *SELECT ELEMENTS AND PRESS THE RIGHT BUTTON TO CONTINUE.*

Position the cursor over the elements you want to select and click the mouse button to confirm.

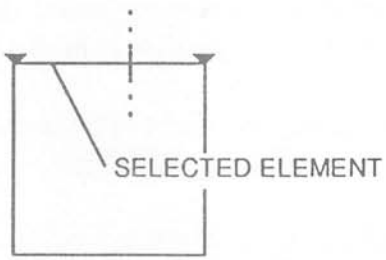
Note the triangular shapes that indicate the selected elements.

You can repeat the selection as many times as you wish. When satisfied, press the right mouse button to return to the drafting board.

To deselect elements, open the **Select** menu and then *Deselect Elements*. You will see the message *SELECT ELEMENTS AND PRESS THE RIGHT MOUSE BUTTON TO CONTINUE,* and a cross-type cursor on the screen.

The deselection process works exactly the same way as the selection process.

Fig. 206



You can separate one element from one object, but if you select this object the element will always be part of it.

REMEMBER: Only selected items may be processed.

OBJECTS

These are sets of lines that make up any figure. They can be selected and processed as a unit. Objects can be created from figures included in the **Tools** menu, or by combining several of the above using the group process feature.

To select, open the **Select** menu and then *Objects*. You will see the message *SELECT OBJECTS AND PRESS THE RIGHT MOUSE BUTTON TO CONTINUE.*

Position the cursor over any side of the object(s) you wish to select and click the mouse button to confirm.

Note the triangular shapes that indicate the selected objects.

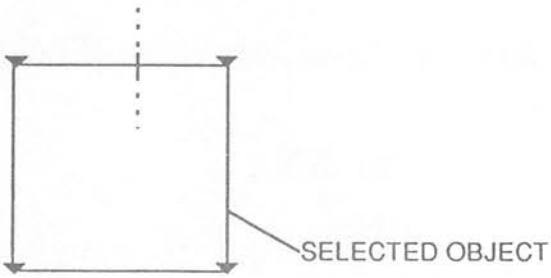
You can repeat the selection process as many times as you wish. When satisfied, press the right mouse button to return to the drafting board.

To deselect objects, open the **Select** menu and then choose *Deselect Objects*. You will see the message *SELECT OBJECTS AND PRESS THE RIGHT MOUSE BUTTON TO CONTINUE,* and a cross-type cursor on the screen.

The deselection process works exactly the same way as the selection process.

You can combine different selection criteria as well. For example, you can select an object, then deselect any points from it, then select an element from another object, etc.

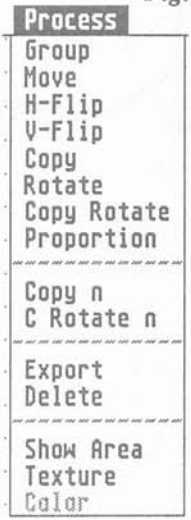
Fig. 207



PROCESS

Menu to choose the process to use on selected items.

Fig. 208



GROUP

Command to group the selected items into one integrated object.

Open the **Process** menu and select *Group*.

REMEMBER: The process options function only on selected items. Unselected (as opposed to deselected) items will not be affected by the chosen process.

MOVE

Command to move the selected items from one user-defined position to another.

Select the items you wish to move, then open the **Process** menu and select Mo*v*e. You will see the message *SET INITIAL BASEPOINT POSITION a*nd a cross-type cursor.

Place the cursor where you wish to define the basepoint, or first point of reference. Click the mouse button to confirm.

You will now see the message *SET FINAL BASEPOINT POSITION*. Move the cursor to where you want the new point of reference to be, and click the mouse button.

You will see that all selected points have moved in the direction marked by the two reference points, also in proportion to the distance between the first point and the second.

H-FLIP and V-FLIP

The Commands to flip the selected items horizontally or vertically on a user-defined axis.

You can flip the items horizontally from right to left or vice versa, depending on the flip axis position. You can either flip the items vertically from top to bottom or vice versa.

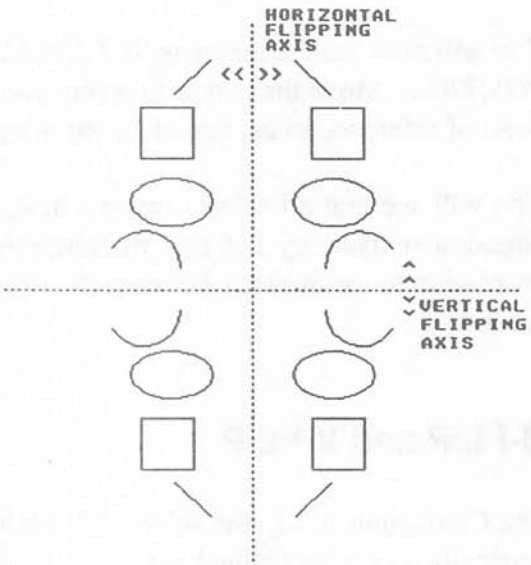
Select the items you wish to flip. To be sure that you have selected them, open the **Select** menu, select *Deselect All,*

*t*hen carefully select only those items you wish affected by the flip process.

Once the items have been selected, open the **Process** menu and (for example) select the *H-Flip* option. You will see the message *SET THE FLIP AXIS a*nd a broken vertical line representing the axis.

Position the cursor where you want the axis to be and click the mouse button to complete the operation and return to the drafting board. Note the changed position of the item selected.

Fig. 209



COPY

Command to copy the selected items from one user-defined position to another.

Select the items you wish to copy, then open the **Process** menu and select *Copy*. You will see the message *SET INITIAL BASEPOINT POSITION a*nd the cross-type

cursor.

Place the cursor over the point where you wish the base-point to be, and click the mouse button to confirm. You will now see the message *SET FINAL BASEPOINT POSITION*.

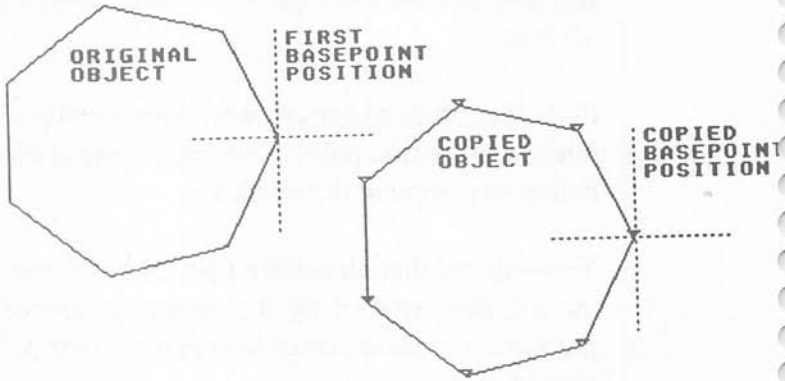
Place the cursor where you want the copied position to be, relative to the first point of reference, and click the mouse button to complete the operation.

You will see that all selected points have been copied to the location marked by the second reference point, in proportion to the distance between the first point and the second.

> ***Note: For the purposes of this process, all copied elements will constitute an integrated object. As before, only selected elements will be copied.***

The new object will appear selected after the copying process is terminated.

Fig. 210



ROTATE

Command to rotate the selected items in respect to a user-defined axis.

The rotation is made from one user-defined angular position to another.

Select the item to be rotated, open the **Process** menu and select ROTATE. You will see the message *SET SPIN CENTER* and the cross-type cursor.

Position the cursor where you want to define the rotation center and click the mouse to confirm.

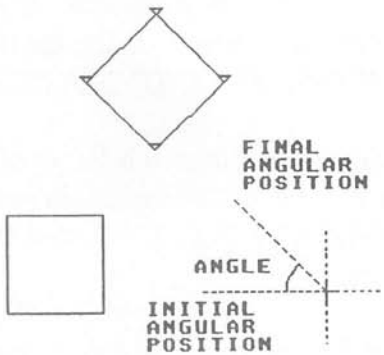
See the dialog box indicating the options for defining the rotation angle by numeric input or graphic mode. For the first option, simply enter the desired angle value and click on *OK* to confirm. In the second case, select *Graphic Mode* and click on *OK* to confirm.

Move the mouse, and note the indicator rotating around the fixed point and the message *SET INITIAL ANGULAR POSITION*. Position the indicator to set it and click the mouse to confirm. Now you will see the message *SET FINAL ANGULAR POSITION a*nd the indicator will appear once again.

Now set the new angular position and click the mouse button to complete the rotation.

To be rotated, an object must have all of its points selected.

Fig. 211



COPY ROTATE

Command to simultaneously copy and rotate an item or items in respect to a user-defined axis.

Copy rotation is done from one user-defined angular position to another.

Select the item first, then open the **Process** menu and select Ro*tate*. You will see the message *SET ROTATION CENTER* and the cross-type cursor.

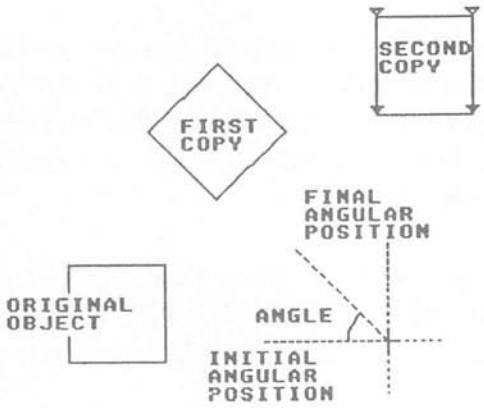
Position the cursor where you want the rotation center to be, and click the mouse to confirm. Move the mouse, and note the indicator rotating around the fixed point and the message *SET INITIAL ANGULAR POSITION*.

Position the indicator to set the degree of rotation and click the mouse to confirm. You will now see the message *SET FINAL ANGULAR POSITION,* and the indicator will appear again.

Now place the indicator in the new angular position and click the mouse to complete the copy rotation.

To be copy rotated, an element must have all points selected.

Fig. 212



COPY n

Command to copy the selected items *n* times. Each copy is made from one user-defined position to another.

Select the items to be copied, open the **Process** menu and select *Copy n*.

You will see the message *SET INITIAL BASEPOINT POSITION a*nd the cross-type cursor.

Place the cursor over the point you wish to define as the basepoint and click the mouse button to confirm.

You will see the message *SET FINAL BASEPOINT POSITION*. Place the cursor where you wish to define the final basepoint. Click the mouse button to complete the operation.

You will see a dialog box that lets you select how many copies you want, and, optionally, the relative perpendicular distance between each copy.

Select *DW a*nd notice that you are working on a view perpendicular to the view you were working on before. Now you will see the message *SET INITIAL BASEPOINT* and a level indicator that lets you establish the first reference level.

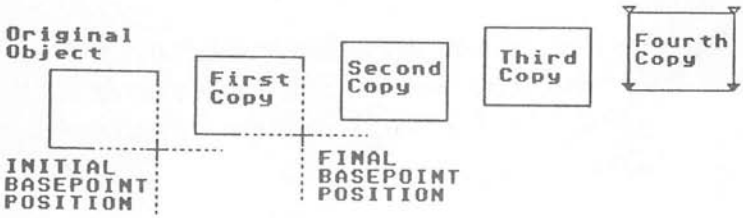
Move it to where you want and click the mouse button. The message *SET FINAL BASEPOINT* will appear, as well as the indicator to set the second reference level.

Move it to the desired position and click the mouse button to confirm. You will then return to a dialog box where you can start over again if you wish.

You will see that all selected points have been copied to the location marked by the second reference point, in proportion to the distance between the first point and the second.

Note: For the purposes of this process, all copied elements will constitute an integrated object. As before, only selected elements will be copied. The new object will appear selected after the copying process is terminated.

Fig. 213



COPY ROTATE n

Command copies and rotates *n* times the selected items in respect to a user-defined axis.

Each item affected by this process is copied and rotated from one user-defined angular position to another. The relative angular difference from each copy to the next is the same as that of the original to the first copy.

Select the item, open the **Process** menu and select *Rotate*. You will see the message *SET ROTATION CENTER* and the cross-type cursor.

Position the cursor where you want the rotation center to be and click the mouse button to confirm. Move the mouse, note the indicator rotating around the fixed point, and the message *SET INITIAL ANGULAR POSITION.*

Position the indicator where you wish and click the mouse button to set it. Now you will see the message *SET FINAL ANGULAR POSITION* and the indicator will appear again.

Move the indicator to the new angular position and click the mouse to confirm. You will see a dialog box that lets you select how many copies you want and, optionally, the relative perpendicular distance between each copy.

Select DW and notice that you are working on a view perpendicular to the view you were working on before. Now you will see the message *SET INITIAL BASEPOINT* and a level indicator that lets you establish the first reference level. Move it to where you want and click the mouse button.

The message *SET FINAL BASEPOINT* will appear, as well as the indicator to set the second reference level. Move it to the desired position and click the mouse button to confirm. You will then return to a dialog box where you can start over again if you wish.

If you are satisfied with the results, select *OK* and you will see the copy rotated and the objects developing on the screen.

To be copy rotated, an element must have all its points selected.

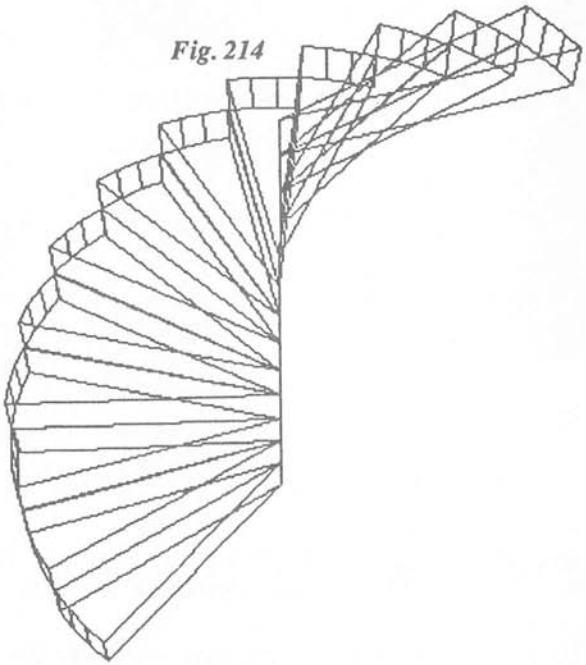


Fig. 214

EXPORT

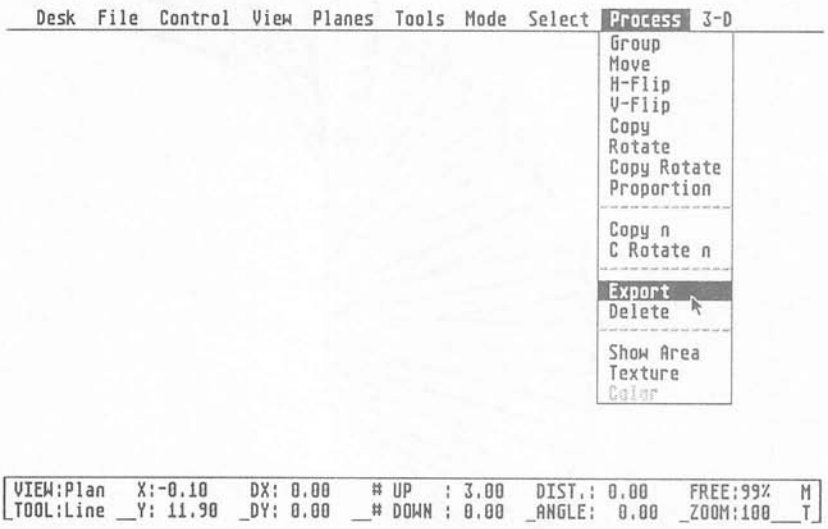
Command to export the selected items to another file.

For example, you may have a need to import elements or items in a certain order to be able to process them properly.

Open the **File** menu and open your "library" file. Select any object, open the **Process** menu and select *Export*.

Now open the file you want to work with by opening the **File** menu and selecting the *Import* option. Notice the objects imported from the "library" file on the center of the screen. Observe that they are selected: this is to let you move and position them wherever you want.

Fig. 215



You can always check on the exported items by opening the CONTAINER.OBJ file.

DELETE

Command to delete the selected items.

Select the items you want to delete. Open the **Process** menu and select the *Delete o*ption. You will see a dialog box warning you about what you are about to do. Select *OK* to delete the items. You can also *Cancel* if you wish, and nothing will happen.

Be careful when using this command!

SHOW AREA

Command to show the area of a defined polygon. A defined polygon simply means that the polygon has been defined; that is to say, it exists.

For the purposes of ***Master CAD,*** a polygon exists if you can see it in *Filled Planes*. There are two ways to be sure that a polygon is measurable:

Previous control: Drawing the polygon in *Contain* mode.

Checking: Seeing that the polygon has texture in *Filled Planes* mode.

To show the area of a defined polygon, select all its points, open the **Process** menu and select *Show Area.*

You will see the message *PRESS LEFT BUTTON TO CONTINUE, RIGHT BUTTON TO EXIT*. You will also see the area of the polygon in the monitor.

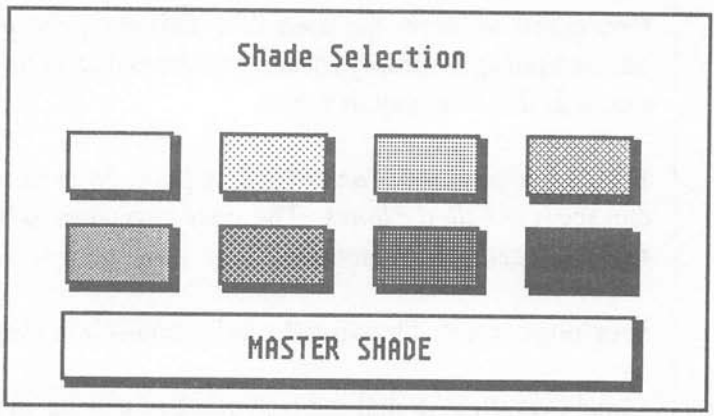
Press the right mouse button to return to the drafting board.

TEXTURE

Command selects the texture of a defined polygon for *Filled Planes* purposes.

To select the texture of a defined polygon, select all their points, open the **Process** menu and select T*exture*. You will see a dialog box with many texture options. Select the texture you want and the system will return automatically to the drafting board.

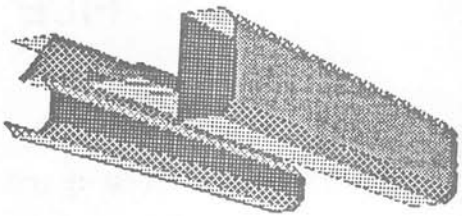
Fig. 216 Fill Pattern Select Box



If you want to check the texture changes, open the **3-D** menu, select an appropriate viewpoint from which to see the polygon, and open the *Filled Planes* viewmode.

Let's continue to:

Fig. 217



3-D MAIN MENU

FILE

Menu to select file operating system activities.

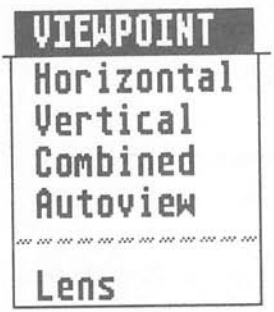
Fig. 218

Functions are the same here as in the 2-D FILE Menu. Refer to page 163.

VIEWPOINT

Menu to select the conditions and parameters of the viewpoint.

Fig. 219



HORIZONTAL

Command to define the observer's position and its horizontal viewing angle.

Create any object in the 2-D section. Open the **3-D** menu and note how the menu bar changes.

Open the **Viewpoint** menu and select *Horizontal*. Notice that you are now working in *Plan* view. You will see the message *SET HORIZONTAL VIEWPOINT* and the cross-type cursor.

Position the cursor where you want and click the mouse to define the horizontal viewpoint. Move the mouse and you will see the rotating line that indicates the viewing direction, or view angle. You will see the message *SET HORIZONTAL VIEW ANGLE.*

Define the view direction you want to use and click the mouse to confirm. You will see the new perspective developing on the screen.

VERTICAL

Command to define the observer's elevation and vertical viewing angle.

Create any object in the 2-D section. Open the **3-D** menu and note how the menu bar changes.

Open the **Viewpoint** menu and select *Vertical.* Notice that you are now working in *Front* view. You will see the message *SET VERTICAL VIEWPOINT* and the level-type cursor.

Position the cursor where you want and click the mouse to define the vertical viewpoint (the height of the viewpoint). Move the mouse and you will see the rotating line that indicates the viewing direction, or view angle. You will see the message *SET VERTICAL VIEW ANGLE.*

Define the view direction you want to use and click the mouse to confirm. You will see the new perspective developing on the screen.

COMBINED

Command to combine the *Horizontal* and *Vertical* viewpoint functions.

When you use this command the program will ask you to set the horizontal viewpoint first, then the vertical viewpoint. It will then develop the new perspective or parallel projection on the screen.

AUTOVIEW

Command that allows you to obtain an autocentered view of the object you are working on.

Open the **Viewpoint** menu and select *Autoview*. You will observe the perspective of the object developing on the screen. Naturally, you will have to create the object in 2-D before using this command.

LENS

Command modifies the viewangle width.

The ***Master CAD*** camera has three lenses you can choose from by opening the **Viewpoint** menu and selecting *Lens*. You will see a dialog box with the following lens options:

NORMAL: This is the default lens and is equivalent to a 55mm lens.

WIDE ANGLE: Equivalent to a 28mm lens.

NARROW ANGLE: Equivalent to a 75mm lens.

Fig. 220 Select Narrow Angle...

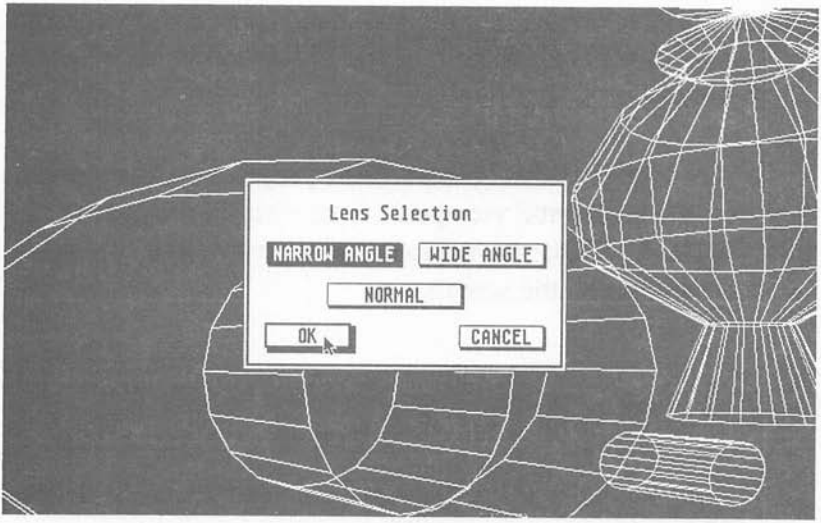


Fig. 221 ...to tighten the view

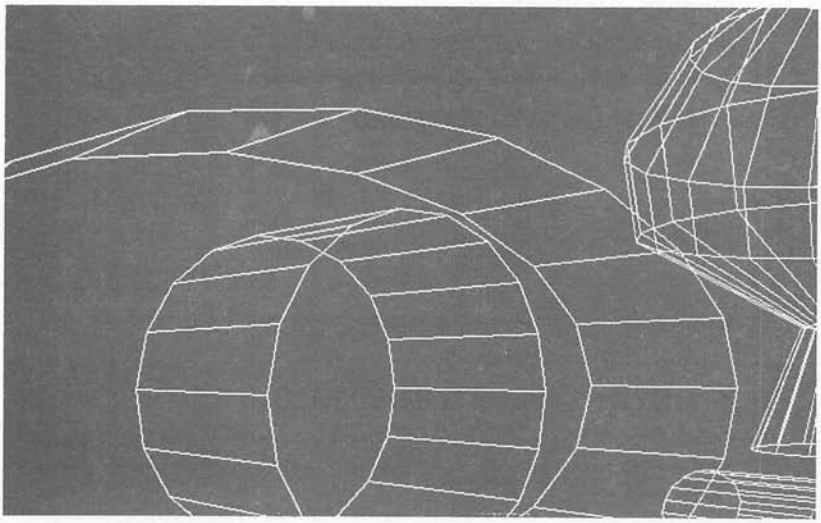


Fig. 222 or Wide Angle...

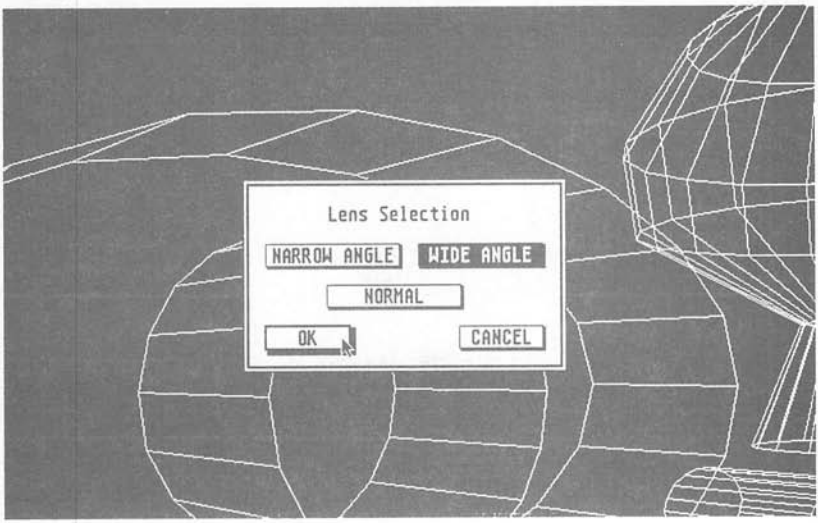
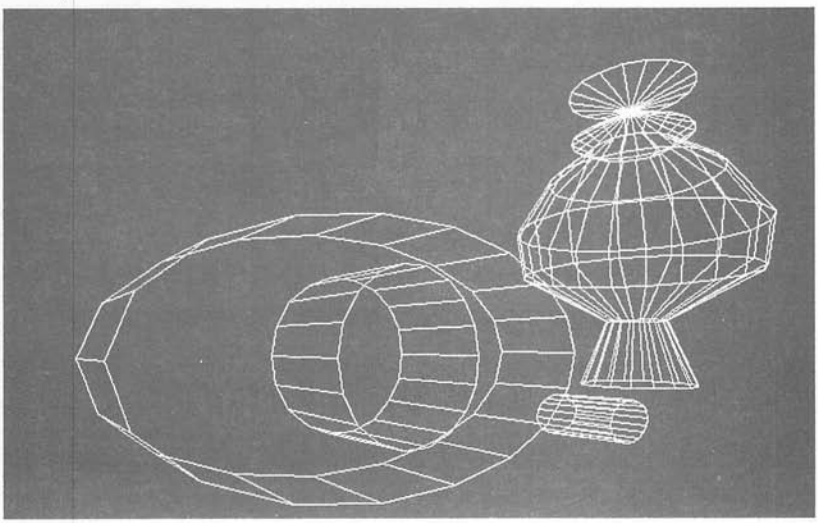


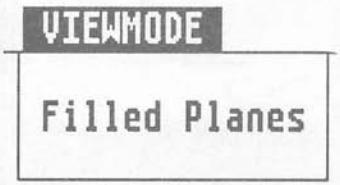
Fig. 223 ...to expand it!



VIEWMODE

Menu activates the solid presentation of the objects.

Fig. 224



FILLED PLANES

Once you have created an object and have seen it in 3-D wire frame mode, you can see it as a solid by opening the **ViewMode** menu and selecting *Filled Planes*.

Fig. 225 Select **Filled Planes**

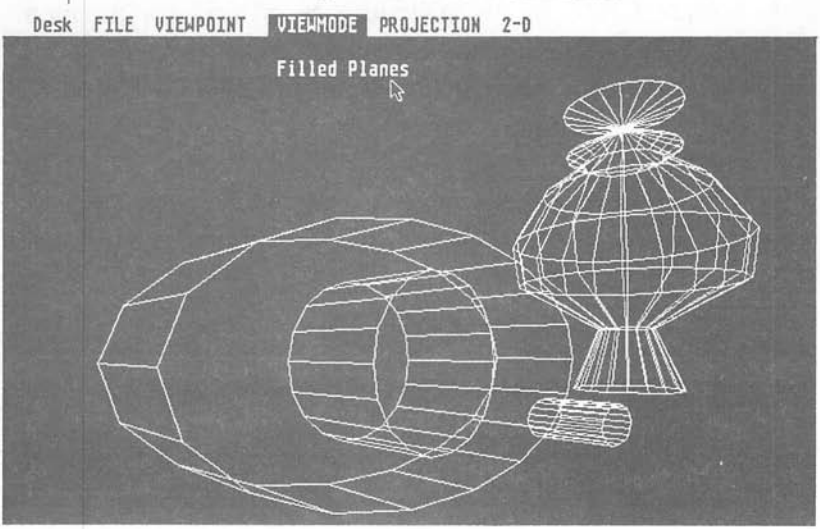
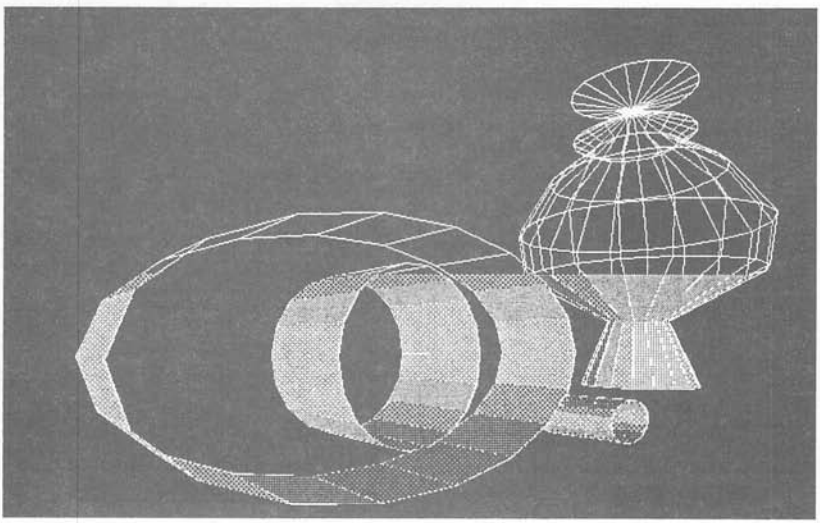


Fig. 226 Patterns fill the Object Planes



Master CAD has its own texture criteria for a given polygon. If you do not like the texture selected by the system, you can change it by using the TEXTURE process in the 2-D section.

Fig. 227 To change fill textures

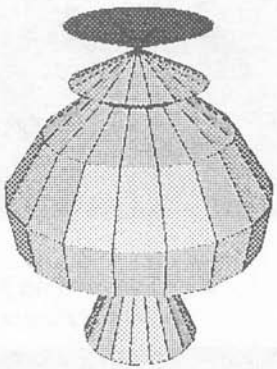


Fig 228 Select the object, then Texture

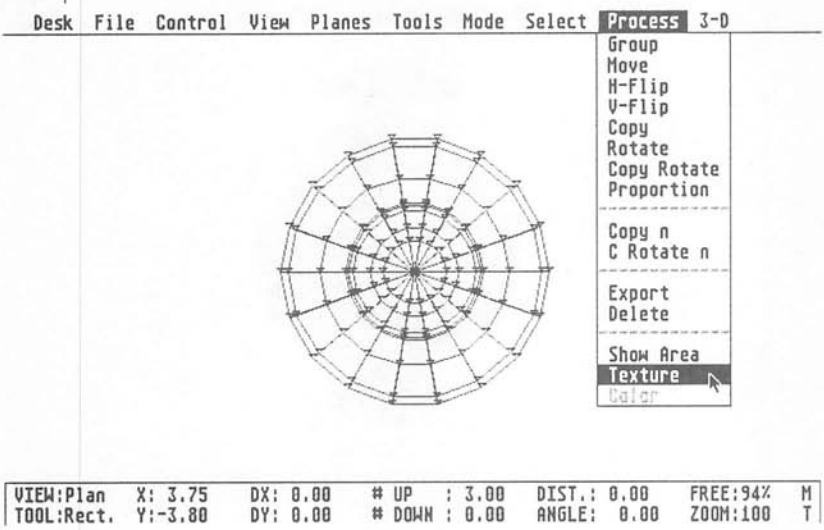


Fig 229 Select the desired texture

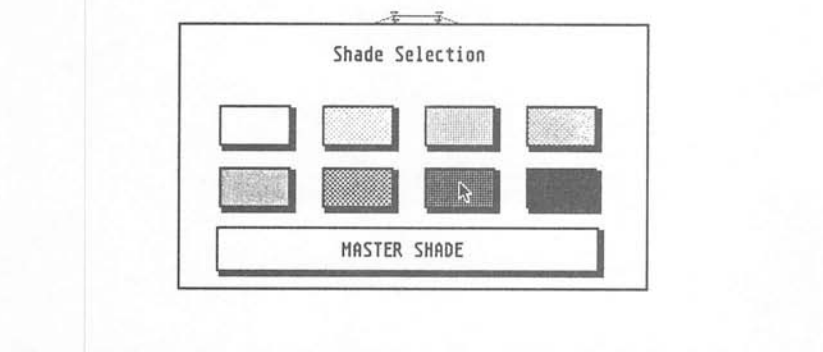
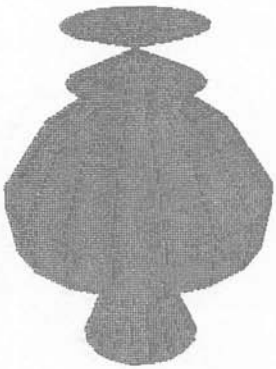


Fig 230 And observe the results (Textures can also be changed for individual Object parts)



PROJECTION

Menu to define the projection in which the object will be represented.

PERSPECTIVE

The object is represented as it would appear in 3-D real life. The effect is like a photograph taken from a given viewpoint. This is the default mode. If you are in *Parallel* mode, open the **Projection** menu and select *Perspective*. You will see the objects developing on the screen.

PARALLEL

The object is represented in 3-D or 2-D depending on user-defined viewing angle. All parallel lines in real life will be represented parallel in this mode.

For example, if you want to obtain an oblique (45-degree) projection of an object, open the **Projection** menu and select *Parallel*.

You will probably see a 2-D representation of the object.

Now go to the **Viewpoint** menu and select *Combined*.

Trace in your mind an imaginary 45-degree line that passes through the center of the object, position the viewpoint on any point on this line and click the mouse button to confirm. Now set the view direction indicator for 45 degrees (using the *Angle* display in the monitor to help you). Click the mouse to confirm. Repeat the operation with the *Vertical* viewpoint. You will see an oblique projection of the object developing on the screen.

This projection also lets you obtain 2-D views of the object from any point you want. For example, if you want to see the shape of a building you have created, place the horizontal viewpoint facing the side the building and the horizontal view angle perpendicular to that side. Repeat the criteria with the vertical viewpoint and activate the *Filled Planes* command.

You will see the shape of the object developing on the screen.

2-D

Toggle to return to the 2-D main menu.

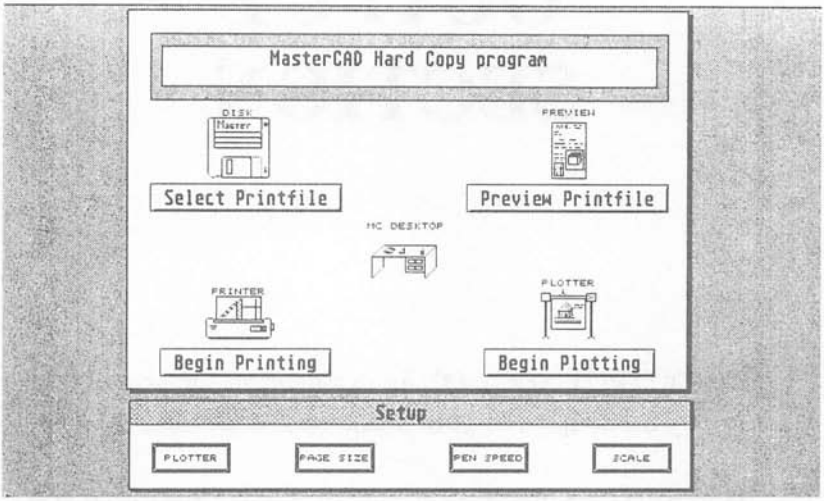
OUTPUT SECTION

The OUTPUT SECTION is designed to let you obtain quality hardcopies of your work on printers and plotters.

You can access this section from the *GEM Desktop* (if you are working on the DESIGN SECTION then open the **File** menu and select the *Quit* option, ***Master CAD*** will bring you to the *Desktop).*

From here select OUTPUT.PRG. The following dialog box will appear:

Fig 231



SELECT PRINTFILE

Command to select the ***Master CAD*** PRINTFILES that exist on disk.

These are special ***Master CAD*** files that you saved with the Sa*ve Screen* command from the DESIGN SECTION. If you want to print or plot your work, you have to have saved it in this format. These files are saved with a .PRF extender.

To select a Printfile place the cursor over the SELECT PRINTFILE icon and click the left mouse button once. You will see a list of files which have the file extender [.PRF]. You can load a file by typing the filename or by double clicking on the filename from the file directory.

Observe the name of the current Printfile selected on the upper part of the screen.

BEGIN PRINTING

Command prints the selected Printfile.

Check that your printer is properly connected to your computer.

Check that paper is correctly installed in your printer.

Place the cursor over the BEGIN PRINTING icon and click the left mouse button to start the printout.

Master CAD is mainly compatible with the EPSON FX 80 or compatible printers. ***Master CAD*** is connected to the G-DOS operative system so theoretically it is compatible with any driver prepared for this Operative System.

BEGIN PLOTTING

Command to plot the Printfile selected.

Check that your plotter is properly connected to your computer.

Check that paper is correctly installed in your plotter.

Place the cursor over the BEGIN PRINTING icon and click the left mouse button to start the printout.

Master CAD is compatible with the Hewlett Packard™ and compatible plotters. This plotter in combination with ***Master CAD*** produces high quality black and white graphics in one size of drawing media: ANSI A (8 1/2 x 11 in.). Other plotter drivers with more drawing sizes will be released soon.

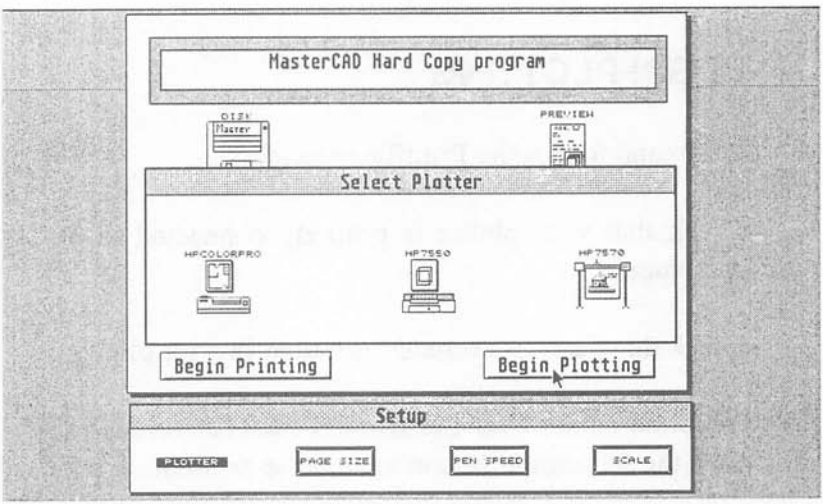
MC DESKTOP

Command returns you to the *GEM Desktop*.

PLOTTER

This command selects the plotter driver to be used. Place the cursor over the PLOTTER icon and click the left mouse button. Observe the *Select Plotter* Window showing the different kind of plotters you can use. Click the mouse over the desired PLOTTER Icon.

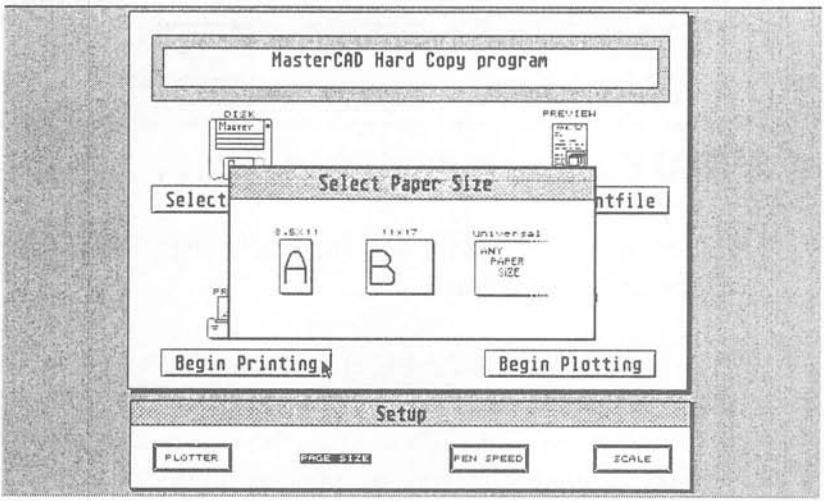
Fig. 232 Select Plotter Type



PAGE SIZE

This command selects the size of the paper to be used with the plotter. Place the cursor over the PAGE SIZE icon and click the left mouse button. Observe the *Select Size* Window showing the different size possibilities. Click the mouse on the desired size icon. If the selected size is not compatible with the previously selected Plotter, an *Alert* Box will tell you.

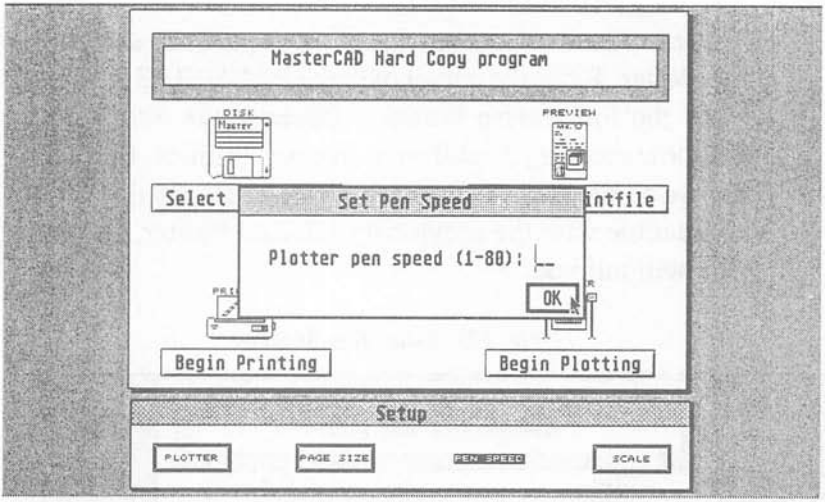
Fig. 233 Select Size Window



PEN SPEED

This command selects the speed of the pen used by the plotter. Place the cursor over the PEN SPEED icon and click with left mouse button. Notice the *Speed Input* Window asking for the desired speed. Type it in using the keyboard. If a PEN SPEED is selected that is not supported by the selected plotter, a warning dialog box will inform you.

Fig. 234 Speed Input Window

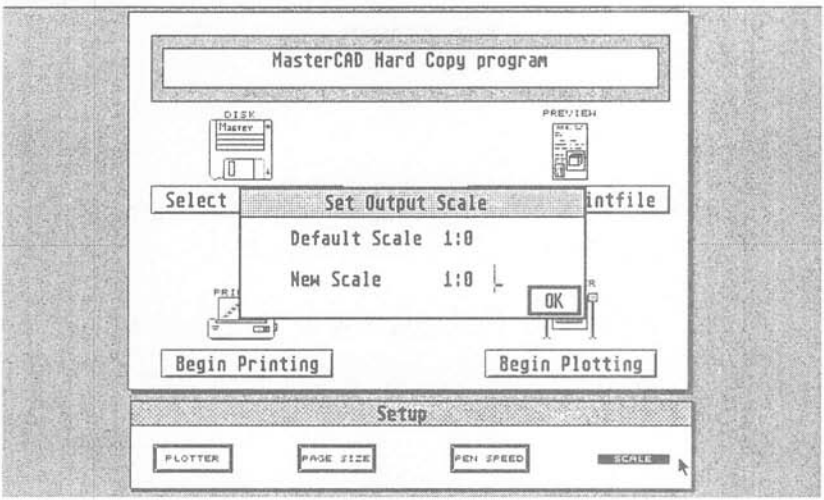


SCALE

This command selects the scale at which the PRINTFILE will be printed or plotted.

Place the cursor over the SCALE icon and click the left mouse button. Observe the *Select Scale* window showing the different possible scales. Click the left mouse button over the desired size icon, or type the input option and press *OK* to confirm.

Fig. 235 Select Scale window



INSTALLING YOUR PLOTTER

This section is designed to give you some examples on how to install your plotter device properly.

Make sure that you have connected your plotter with your computer using a "Null Modem" cable. (7-7, 2-3, 3-2 cable connection).

HEWLETT PACKARD™ COLOR PRO™

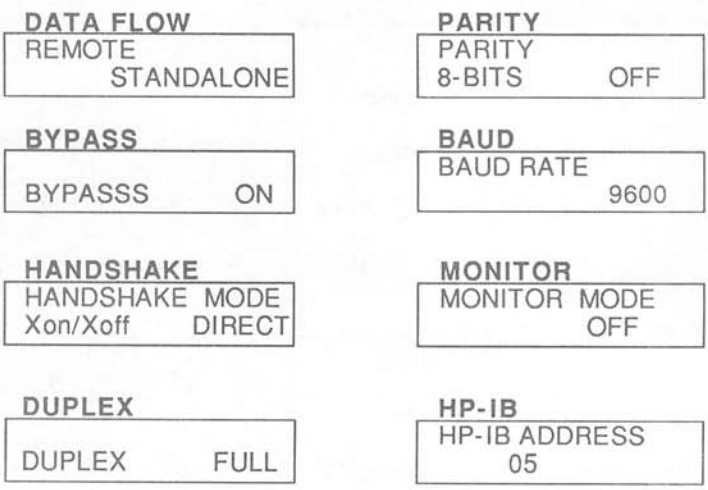
FIG. 236

Dip Switches	I	O
B1		X
B2	X	
B3		X
B4	X	
US		X
S1		X
S2		X

HEWLETT PACKARD™ 7550A GRAPHIC PLOTTER

Displays:

FIG. 237



HEWLETT PACKARD™ 7550A GRAPHIC PLOTTER

FIG. 238

INTERFASE MODE			RS232-C
PARITY	ON	-	**OFF**
	EVEN	-	**ODD**
DUPLEX	HALF	-	**FULL**
	HARDWIRE	-	MODEM
DTR	BYPASS	-	**NORMAL**

HP-IB PORT
COMPUTER/MODEM
TERMINAL
BAUD RATE : 9600 BAUD

EXPAND	-	**NORMAL**
EMULATE	-	**NORMAL**
STANDALONE	-	EAVESDROP
MONITOR MODE	-	**NORMAL**
LOCAL	-	**NORMAL**

NOTE : Use the functions that appear in **bold** style.

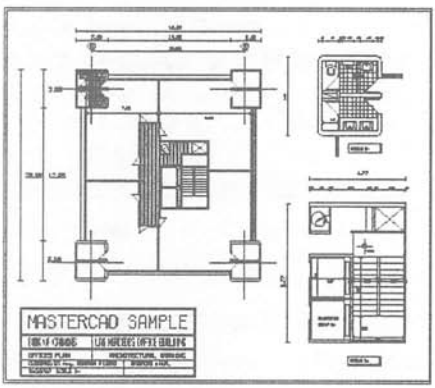
NOTE : All this switches are located on the back of your plotter.

PLOTTING MULTIPLE PAGES

As you know, ***Master CAD*** PRINTFILES can be saved in any scale you desire. In some cases this will cause multiple page plotting. If you are plotting a multiple page PRINTFILE, follow the screen instructions to determine when to insert the next page.

Chapter 5

Reference Charts



2-D MAIN MENU

FILE

Menu to select operating system activities.

NEW OPEN APPEND IMPORT SAVE
SAVE-AS DRAFT-PRINT SAVE-SCREEN QUIT

CONTROL

Menu to select operating system activities.

ZOOM GRID UNITS SAVE
REVERSE-SCREEN REVOLUTION-STEP RULERS ON/OFF

VIEW

Selects the orthogonal view you will work with.

PLAN FRONT BACK LEFT RIGHT

PLANS

Menu to determine the type of projection plane to work with.

NORMAL PLAN-ANGLED FRONT-ANGLED SIDE-ANGLED

TOOLS

Selects the graphic tool you want to use for drafting.

LINE RECTANGLE CIRCULAR POLYLINE
POLYGON Clockwise ARC CounterClockwise ARC TEXT DIM

MODE

Menu to select the projection mode you will work with.

PLANES: CUT HI-CONTAIN LOW-CONTAIN
REVOLUTION: 360 CLOCKWISE COUNTER CLOCKWISE

SELCT

Menu to choose the selection or deselection criteria.

POINTS ELEMENTS OBJECTS ALL

PROCESS

Menu to determine the type of process to use with selected items.

GROUP MOVE FLIP-HORIZONTALLY FLIP-VERTICALLY
COPY ROTATE COPY-ROT PROPORTION COPY-N
COPY-ROT-N EXPORT DELETE SHOW-AREA TEXTURE

3-D

Menu Command to switch to the 3-D Menu (See chart on page 255):

FILE VIEWPOINT VIEWMODE PROJECTION 2-D

3-D

FILE

Menu used to observe and modify an object's perspective.

FILE VIEWPOINT VIEWMODE PROJECTION 2-D

VIEWPOINT

HORIZONTAL: Command to define through cursor movement, the observors position and his horizontal viewing angle.

VERTICAL: Command to define through cursor movement, the observors position and his vertical viewing angle.

COMBINED: Command that combines the function of horizontal and vertical commands.

AUTOVIEW: Command that allows an auto-centered view of an object.

LENS: Commands to modify the viewangle width.

VIEWMODE

Command to activate the solid presentation of objects.

FILLED PLANES

PROJECTION

Menu to define the type of projection that will be used.

PERSPECTIVE: The object in the space is represented as it would appear in 3-D real life. The effect is like a photograph taken from a given viewpoint.

PARALLEL: The object is represented in 3-D or 2-D depending on the viewing angle defined by the user. All the parallel lines in real life will be parallel in this mode.

2-D

Command to return to two-dimensional drawing.

FILE

NEW	Command to reset all work parameters and restart with the default conditions. It also creates a workfile from scratch.
OPEN	Command to open a previously saved [.OBJ] file, it resets all parameters and removes any information contained in RAM.
APPEND	Command to open a previously saved [.OBJ] file and append it to the current file in memeory. It doesn't reset the current parameters. Check the amount of available memory before starting this operation since Master CAD will accept only the part of the appended file that it could fit in the available memory.
IMPORT	Command to import the exported objects from another file. Check the amount of available memory before starting this operation since Master CAD will accept only the part of the appended file that it could fit in the available memory.
SAVE	Command to save to disk the objects created as a FILENAME.OBJ file. If the file is previously saved with a user defined filename this command will save it with this filename.
S-AS	Command to save to disk the objects created with a user defined filename. Example: HOUSE.OBJ
DRAFT	Command to send a SCREEN DUMP to the printer.
S-SCREEN	Command to save the screen contents to different formats. DEGAS FORMAT Example: HOUSE.PI3 MASTERCAD PRINTFILE Example: HOUSE.PRF
Q	Command to quit the DESIGN section and return to desktop.

PROCESS

GROUP	Command to group the selected items as a single unique object.
MOVE	Command to move the selected items from a relative user defined position to a new position also defined through cursor movement by the user.
FLIP-HOR	Command to flip the selected items horizontally, with respect to a user defined axis.
FLIP-VER	Command to flip the selected items vertically, with respect to a user defined axis.
COPY	Command to copy the selected items from a relative user defined position to a new position also defined through cursor movement by the user.
ROTATE	Command to rotate the selected items with respect to a user defined axis. The rotation is made from the user selected angular position to a new angular position also determeined by the user.
COPY-R	Command to copy-rotate the selected items with respect to a user defined axis. The copy-rotation is made from the user selected angular position to a new angular position also determined by the user.

PROCESS

PROPOR	Command to modify the proportion of the selected items. The proportions vary with respect to a user defined center.
COPY-N	Command to copy "N" times the selected items from a relative user defined position to a new position also defined through cursor movement by the user. The copy follows the relative sequence between them determined by the user.
COPY-RN	Command to copy-rotate "N" times the selected items with respect to a user defined axis. The copy-rotation is made from a user selected angular position to a new angular position also determined by the user. The copy follows the relative sequence between them determined by the user.
EXPORT	Command to export the selected items to another file.
DEL	Command to delete the selected items.
AREA	Command to show the area of a selected polygon.
TEXTURE	Command to select the texture for a selected polygon. It works only for filled plane purposes.

CONTROL

ZOOM

Command to modify the working zoom. The range is 1 to 1000.

GRID

Command to set the working grid parameters.

GRID ON/OFF SNAP TO GRID SET GRID

UNITS

Command to select the measurement units.

INCHES METERS CENTIMETERS MILLIMETERS

ORIGIN

Command to indicate a user defined coordinate origin point. It works through cursor movement.

REVERSE

Command to reverse the screen from white to black and vicev-ersa.

CENTER

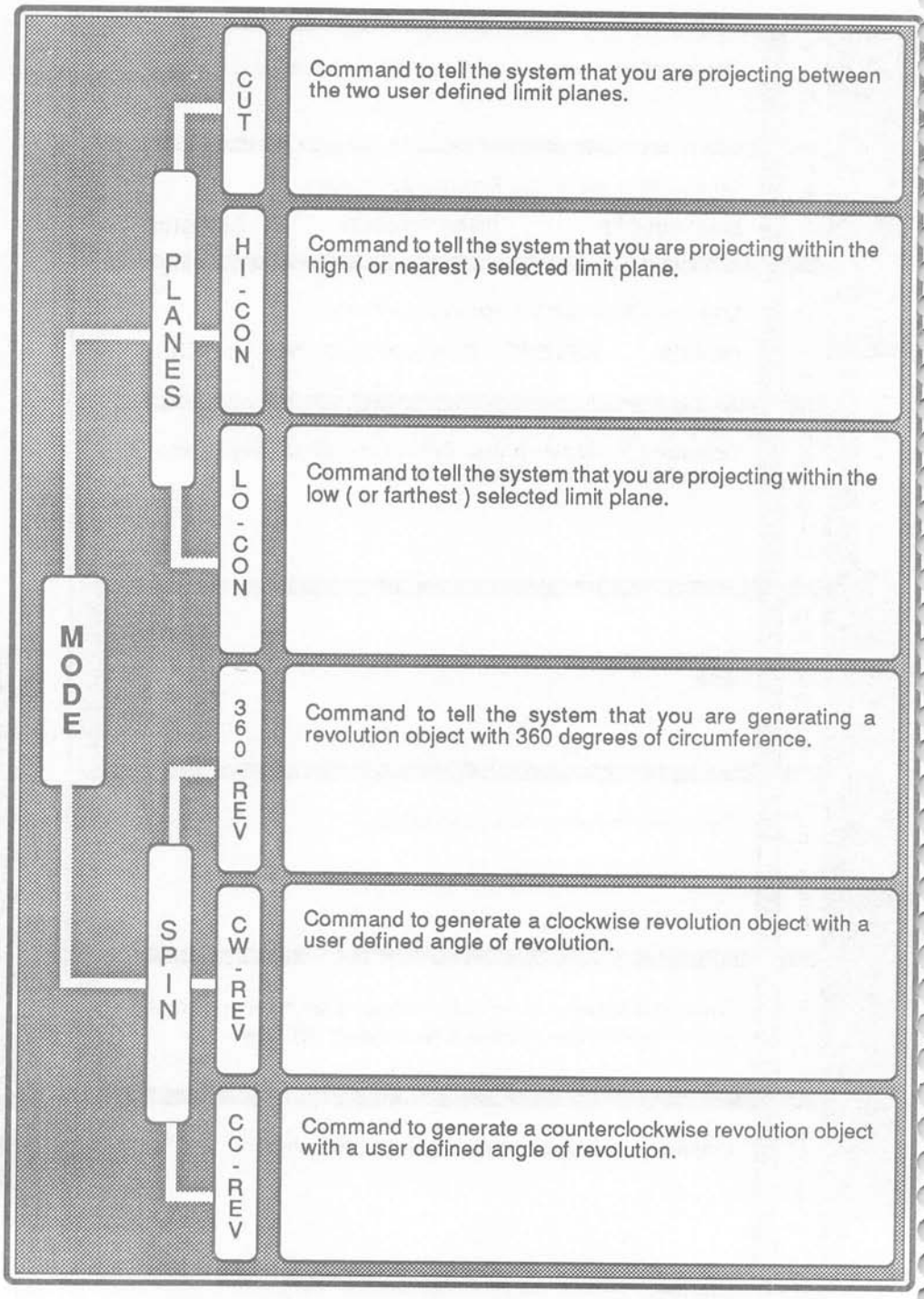
Command to define the screen center.

REVOL

Command to select the circular polygon step number, in other words, the number of sides a spun object will have.

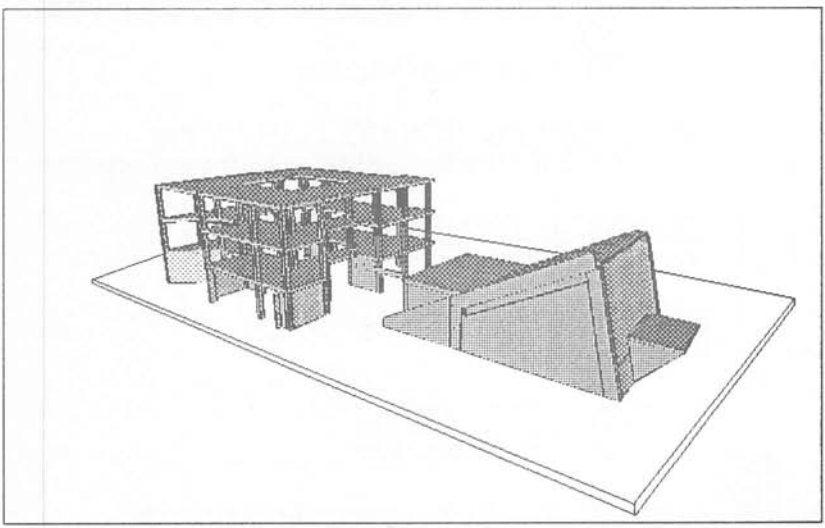
RULER

Command to switch on and off the graphic rulers.


MODE
PLANES
CUT
Command to tell the system that you are projecting between the two user defined limit planes.
HI-CON
Command to tell the system that you are projecting within the high (or nearest) selected limit plane.
LO-CON
Command to tell the system that you are projecting within the low (or farthest) selected limit plane.
SPIN
360-REV
Command to tell the system that you are generating a revolution object with 360 degrees of circumference.
CW-REV
Command to generate a clockwise revolution object with a user defined angle of revolution.
CC-REV
Command to generate a counterclockwise revolution object with a user defined angle of revolution.

Chapter 6

Appendices



APPENDIX A

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Dr. Rafael Marquez

MICROCOMPUTER GRAPHICS
Roy E. Myers

PERSONAL PASCAL
Optimized System Software

PERSPECTIVE a new system for designers.
Jay Doblin

APPENDIX B

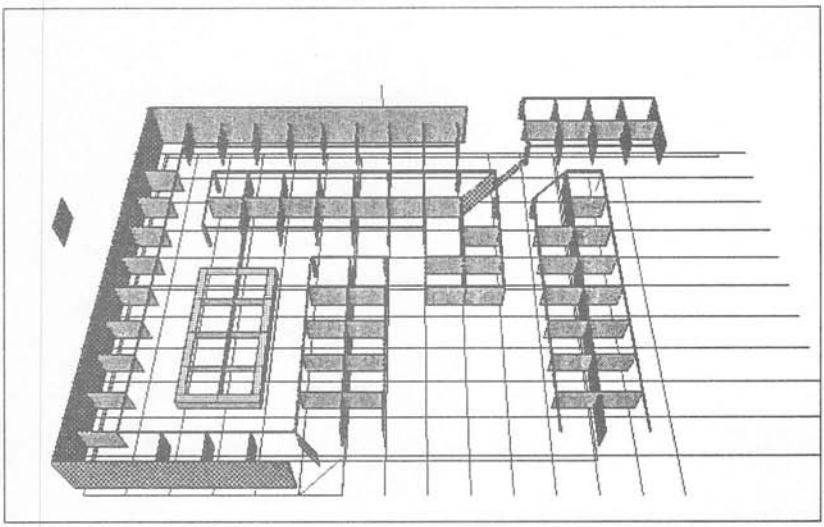
Installing Master CAD on a Hard Drive

To Install the Master CAD system on a hard drive simply copy the disk contents into a folder in the C partition. You can do this by inserting the MasterCAD disk into drive A: then dragging the Disk Icon into the specified Hard disk Partition. You can also drag the files individually (See your Atari Manual for information on copying Disks and files).

Master CAD can be installed in a folder, but the files COLOR.RSC and MONO.RSC must remain in the root directory of the partition.

Make sure GDOS is properly installed in the AUTO folder of the Hard Drive. Also, as the program uses GDOS for its output program, the AUTO folder and all files that have a .SYS extension *must* be copied into the root directory of Drive C.

Index



S

Notes

Notes

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The Roundtable Special Interest Groups (SIG) gives you a means of conveniently obtaining news about our current products, new releases, and future plans. Messages directly from the authors give you valuable technical support of our products, and the chance to ask questions (usually answered within a single business day).

GEnie differs from other computer communication networks in its incredibly low fees. With GEnie, you don't pay any hidden charges or minimum fees. You pay only for the time you're actually on-line with the MICHTRON product support Roundtable, and the low first-time registration fee.

For more information on GEnie, follow this simple procedure for a free trial run. Then if you like, have ready your VISA, Mastercard or checking account number and you can set up your personal account immediately — right on-line!

1. Set your modem for half duplex (local echo)—300 or 1200 baud.
2. Dial **1-800-638-8369**. When connected, type **HHH** and press **Return**.
3. At the **U#=** prompt, type **XJM11957,GENIE** and press **Return**.

And don't forget, MICHTRON's Bulletin Board System, The Griffin BBS, is still going strong (the griffin is the half-lion/half-eagle creature on our logo). Our system is located at MICHTRON headquarters in Pontiac, Michigan. For a trial run, call (313) 332-5452.

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