
BlueCAD
Ready for OS/2 WARP
CADWARE

Second edition (November 1996)

BlueCAD

User Guide

CADWARE

Particular Information

Information for contacting CadWare.

To register your product, use the registration form enclosed with BlueCAD. Registered users can receive support and further information about CadWare products.

To contact CadWare write to following address:

CadWare S.r.L.
Via Roma 55 Noventa Padovana
35027 Padova
Italy

Tel: **049-8932222**
Fax: **049-8932561**

E-mail:

Internet: **cadware@ cadware.it**
CompuServe: **100136,3520**

At the CadWare' WWW internet site:

http\\www.cadware.it
you can obtain technical support, general and technical information, and update news about BlueCAD and the other CadWare products.

Index

Introduction

What is BlueCAD

BlueCAD is a *CAD* (Computer Aided Design) product for the *operating systems* OS/2 Warp, Windows 95, NT BlueCAD applies to a professional and non-professional use, therefore it is not necessary to have experience to use CAD tools.

A fundamental characteristic of BlueCAD is the easy use that enables brief learning time and an immediate individual productivity. This characteristic is due to a simple and accurate graphic interface and operative methods integrated with operative system . The extensive use of *drag&drop*, the presence of *contextual commands*, the personalization of the work environment, and a complete on-line assistance are all characteristics that make the use of BlueCAD simple and intuitive.

BlueCAD supplies the drawing tools necessary for professional use. BlueCAD allows one to create, modify, transform, and manage a complete set of geometrical objects without having limitations on the number of graphic entities usable in a drawing. *Associative dimensioning*, management of *layers* and *blocks*, coupling with the *snap points* of the graphic entities, *grid*, an integrated calculator with a formula interpreter, and a *plotting* (OS/2 Warp) module are some of the main instruments made available by BlueCAD. The functionalism of reading/writing of the *DXF* format, management of *bitmap*, network support, the use of the *REXX* (OS/2 Warp), or *C\C++* (Windows) language for the customization of the system, united with OS/2's own *multitasking*, give BlueCAD the modularity and integrability requisites for the passage from drawing to project and from individual use to use in productive structures.

The OS/2's characteristics, united with BlueCAD's own methods of data processing, make BlueCAD's services possible in terms of speed.

The BlueCAD Package

The BlueCAD package contains the following material:

- 3 disks containing the program or a CD-ROM.
- The BlueCAD registration form.
- The BlueCAD warranty.
- This User Guide

BlueCAD documentation

The sources of information on the use of BlueCAD are:

- This User Guide. By reading the guide and applying what you've learned, you will know and use the main BlueCAD functions. The step-by-step development of a project carried out by BlueCAD permits learning the fundamental techniques necessary for a more efficient and productive use of BlueCAD.
- The on-line *Reference Guide* is the means for quickly finding information while using BlueCAD. It provides a syntactical and semantical explanation of the commands, a detailed description of the graphic interface elements, and also an illustration of the fundamental procedures of BlueCAD.
- The file *Read.me*. This file contains general information and last minute information not yet reported on other documentation.

How to learn to use BlueCAD

The purpose of this User Guide is to lead to the learning and use of BlueCAD, illustrating the main characteristics, functions and procedures.

Reading this User Guide is essential for first-time BlueCAD users. The User Guide illustrates the use guide lines of BlueCAD in the context of practical exercises. The purpose is to provide a basic methodology for a personal and efficient use of BlueCAD. Therefore the expert user should consult the on-line *Reference Guide* for detailede descriptions of all the product`s commands and functions.

The User Guide is made to be read entirely and in order of the chapters as a first approach to BlueCAD, rather than for refering to it when one has

already developed a certain knowledge of the product. It is not necessary to be familiar with CAD or the operating system to consult this User Guide. Furthermore, if there are unknown terms, expressions and concepts during the reading, one can always refer to the *Glossary* located at the end of the User Guide. The best way to learn to use the product is by reading this User Guide while using BlueCAD, putting acquired information into practice as one goes along. In the first consultation one can omit reading the notes.

The First Part, *"The Basics"*, leads the user to the first approach with BlueCAD, from the installation to the creation of simple drawings. *Chapter 1* illustrates the installation and maintenance (reinstallation, uninstallation, updating) of BlueCAD. *Chapter 2* familiarizes the user with the graphic interface of BlueCAD and with the methods of interaction with the program. *Chapter 3* teaches the fundamental drawing operations, developing a simple drawing as an example. You learn as the drawing and the modification of the graphic primitives are executed, using some of the fundamental commands, carrying out save and print procedures.

After these preliminary and fundamental three chapters, in the Second Part, *"Drawing techniques"*, you gradually learn drawing techniques of a plan through a guided step-by-step development. In *Chapter 4* you begin to lay out the work area and the general characteristics of the project, to then be able to see some creation and modification operations of *graphic primitives*. By analyzing the drawing process you learn the techniques and instruments made available. *Chapter 5*'s focus is on the transformation of the graphic primitives and the use of the *views*. In *Chapter 6* you learn to define and modify the attributes of the graphic primitives and to obtain information regarding drawing and primitives. *Chapter 7* illustrates the use of blocks and layers, CAD's specific instruments and fundamental organization elements of the drawing. *Chapter 8* details printing and plotting.

The third and final part of the User Guide, *"Advanced drawing techniques,"* is specifically reserved for advanced drawing instruments. *Chapter 9* is dedicated to the use of dimensions. *Chapter 10* and *Chapter 11* are dedicated to the passage from drawing to project. In particular, the methods of information exchange between BlueCAD and other programs is explained. *Chapter 11* is dedicated to the BlueCAD customization.

A *Glossary* is presented at the end of the User Guide. The *Glossary* gives the explanation of CAD's specific terminology and concepts.

Conventions and terminology

This User Guide uses the following conventions:

- The different section of operative system are identified by simbols:

Windows 95, NT

OS/2 Warp

- The names of the items, commands and keys to be selected and also the graphic interface elements of BlueCAD are in **boldface** type. The combinations of keys are indicated by interposing the symbol + among the symbols of the keys. For example **Alt+F1** indicates the contemporary pressing of the two keys **Alt** and **Fl**.
- Terms and expressions from the *Glossary* are in *italics* when they are introduced, as are references to titles of paragraphs or chapters and variable information that must be substituted by an effective value.
- The text and the information visualized on the BlueCAD screen are indicated in `fixed spacing` charcters.
- The names of the files, paths and commands of the operating system are in CAPITAL letters.
- An x,y coordinate point is indicated by the notation (x,y). Therefore (10,20) means the coordinate point x=10 and y=20.

The following terms are used to describe the actions of the mouse:

Selection, choose or click: point to an element and click mouse button 1.

Open and click twice: point to an element and click twice mouse button 1.

Highlighting: point to an element with the key to move the cursor or using mouse button 1.

Drag and drop of an object: the following operation is meant by this expression:

OS2WARP

1 Position the pointer of the mouse on an object.
2 Hold down mouse button 2.
3 Move the object towards the desired point, by dragging the same object in the desired direction.
4 Release mouse button 2 to bring about the release of the object.

1 Position the pointer of the mouse on an object.
2 Hold down mouse button 1.
3 Move the object towards the desired point, by dragging the same object in the desired direction.
4 Release mouse button 2 to bring about the release of the object.

First Part - The Basics

This first part, "*The Basics*", takes the user through the introduction to BlueCAD, from installation to the creation of simple drawings*Chapter 1* illustrates the installation and maintenance (reinstallation, uninstallation, updating) of BlueCAD.*Chapter 2* familiarizes the user with the graphic interface and the concepts and methods of interaction with the program. *Chapter 3* teaches the basic drawing operations, developing a simple drawing as an example: you learn as the drawing and the modification of the graphic primitives are executed, using some of the fundamental commands, and carrying out save and print procedures.

Chapter 1. Installing BlueCAD

Preliminary operations

Before installing BlueCAD, it's a good idea to carry out some procedures and controls.

- Before every other operation make a reserve copy of the BlueCAD diskettes, keeping the originals in a safe place.
- Carefully read the READ.ME file that is found in the installation disk 1. Recent information that still hasn't been documented elsewhere is brought into this file.
- The computer in which BlueCAD is installed has to satisfy the following requisites:

  - ***Operating system:*** OS/2 Warp, Windows 95 or Windows NT.
  - ***Computer:*** based on Intel 386 or superior processor.

    Note: If you have a 386 processor it's necessary to install a 80387 mathematical coprocessor or an equivalent one.
  - ***RAM Memory*****:** 8 MB are required.
  - ***Hard disk unit*****:** the installation of BlueCAD requires around 9 MB of free space on hard disk.

    Note: Once BlueCAD is installed, regular checking that the free space on the disk is sufficient for current activities is recommended.
  - ***Video card*****:** any video card compatible with the operating system.
  - ***Diskette*****:** a minidisk unit of 1.44 MB.
- ***Mouse*****:** any mouse compatible with the operating system.

- Before installing BlueCAD check that:
  - the operating system is installed in the computer in use.

  - the *printer* or the *plotter* that you wish to use with BlueCAD is installed. BlueCAD supports all the printers compatible with the operating system in use and the plotting formats HPGL and CALCOMP.

    Note: If the operating system, the printer or the plotter haven't already been installed, refer to the operating system documentation for installation instructions.

Installation procedure

To install BlueCAD do the following operations:

Installing BlueCAD from diskettes.

1. Insert the minidisk 1 in unit A.
2. At the command prompt type A:\INSTALL, then press the **Return** key.
3. Follow the instructions on the screen. When the window **Install - directories** appears, if you want to change the suggested C:\BLUECAD installation directory, enter the desired directory in the entry field. Select **Disk space** if you want to see which partition has enough space for the installation and to change the installation partition.
4. Push the **Install** button to initiate the transfer of the BlueCAD files on the installation disk. Follow the instructions, inserting the installation minidisks when required.
   Note: Installation can be interrupted at any moment by selecting the **Stop** pushbutton: in this case the program gives the possibility to cancel the already transfered files from the installation disk.

5. Upon complete installation, select the **Exit** pushbutton, remove the installation minidisk from unit A and restart the computer.


Installing BlueCAD from the CD-ROM.

1. Insert the CD-ROM in the CD-ROM drive. Execute the following procedure (where "**x:**" is the drive letterof your CD-ROM dirve).
2. At an OS/2 command prompt type **x:** and then press **ENTER** to switch to the CD-ROM drive.
3. Type X:\OS2\INSTENG and press **ENTER**.
4. Follow the instructions on the screen.


Installing BlueCAD from diskettes.

1. Insert the minidisk 1 in unit A.
2. Click the **START** button, then click **Run** from the popup menu.
3. Type A:\SETUP in the **dialog Open field**.
4. Follow the instructions on the screen. When the window **Install - directories** appears, if you want to change the suggested C:\BLUECAD installation directory, enter the desired directory in the entry field. Select **Disk space** if you want to see which partition has enough space for the installation and to change the installation partition.
5. Push the **Install** button to initiate the transfer of the BlueCAD files on the installation disk. Follow the instructions, inserting the installation minidisks when required.

   Note: Installation can be interrupted at any moment by selecting the **Stop** pushbutton.

6. Upon complete installation, select the **Exit** pushbutton, remove the installation minidisk from unit A and restart the computer.

Installing BlueCAD from the CD-ROM.

1. Insert the CD-ROM in the CD-ROM drive. Execute the following procedure (where "**x:**" is the drive letterof your CD-ROM dirve
2. Click the **START** button, then click **RUN** from the popup menu
3. Type X:\WINDOWS\INSTENG in the **dialog Open field**
4. Follow the instructions on screen to complete the installation.

The BlueCAD folder

Once BlueCAD is installed, the BlueCAD folder will be present on the Desktop of your computer, appearing as follows:

The folder is opened by double clicking on it. The folder contains the following objects:

- The *BlueCAD program*

- OS2WARP The utility program *BluePlot*, to manage the printing by means of the plotter.

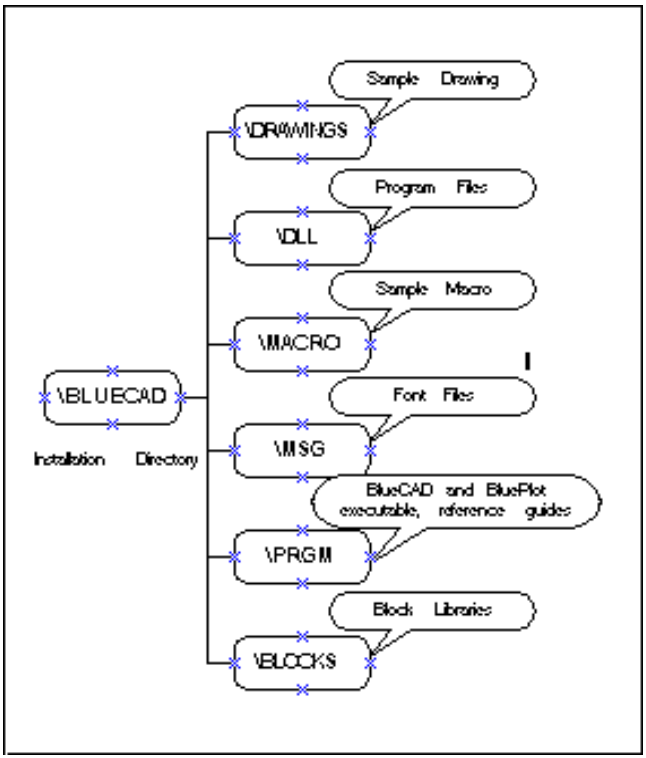
- The BlueCAD on-line *Reference Guide*: it's the means for quickly finding information while using BlueCAD. It provides an explanation of the commands, a detailed description of the graphic interface elements and also an illustration of the fundamental BlueCAD procedures.

- OS2WARP The BluePlot on-line *Reference Guide*: it's the means for quickly finding information while using BluePlot. It provides an explanation of the plotting management program, by means of a detailed description of the options, procedures and graphic interface elements.

- The *Read.me*. file. This file contains last minute news not yet transfered onto other documentation, and also information on how to contact CadWare for information and help.

BlueCAD files and directories

The BlueCAD installation program creates the figure directories in the selected installation partition.



- \BLUECAD is the main BlueCAD directory if a different directory hasn't been chosen in the installation phase. It contains all the other program directories and also the READ.ME file.
- \DRAWINGS is the directory which contains sample drawings files, having .DIS extension. Some of these drawings are utilized in this guide.
- \DLL contains the files having .DLL extension, which are necessary for executing the program.
- \MACRO contains sample macros, which illustrate some of the basic functions implemented by macro.

- \MSG contains the font files and the program messages.
- \PRGM contains the BlueCAD and BluePlot program files, having .EXE extension. It also contains the on-line BlueCAD and BluePlot reference guides.
- \BLOCKS contains the libraries of blocks. The following sample libraries, each with its corresponding directory, are present: architectural blocks (directory \ARCH); furniture blocks (directroy \FURNIT); civil engineering blocks (directory \CIVIL); computer blocks (directory \COMPUTER); blocks used in this guide (directory \GUIDE); electronic logic blocks (directory \LOGIC); mechanical blocks (directory \MECH).

BlueCAD maintenance

Once BlueCAD is installed it is possible to reinstall, cancel or update the present installation by using the same installation program.

Updating

Updating the installation serves for when the user wishes to have a more recent BlueCAD version than the currently installed one. This operation entails the overwriting of the program's files, while the macros and the executed designs found in \DRAWINGS and \MACRO are not cancelled. Carry out the following operations to update BlueCAD:

Updating from diskettes.

1. Insert the minidisk 1 in unit A.
2. At command prompt type A:\INSTALL, press the **Return** key.
3. Follow the instructions on the screen. The program recognizes that a version of BlueCAD has already been installed and proposes the option **Update the currently installed product** in the window **Installation options**. Select **Continue**.
4. Select **Update** in the **Update** window if you want to update the installed product. Otherwise push **Cancel**.

Note: The installation program signals if the version that is being used isn't more recent than the one already installed, giving the possibility to continue or stop the installation.

5. Follow the instructions on the screen, inserting the installation minidisks 2 and 3 when requested.
   Note: If the updating is stopped in progress, selecting the **Stop** pushbutton in the **Update-progress** window will signal that the updating has not been completed, therefore enabling the user to recuperate the previously installed version, hence restoring it.

6. Upon completed updating select **Exit**.


Updating from CD-ROM.

1. Insert the CD-ROM in the CD-ROM drive. Execute the following procedure (where "**x:**" is the drive letter of your CD-ROM dirve).
2. At an OS/2 command prompt type **x:** and then press **ENTER** to switch to the CD-ROM drive.
3. Type X:\OS2\INSTENG and press **ENTER**.
4. Follow the instructions on the screen. The program recognizes that a version of BlueCAD has already been installed and proposes the option **Update the currently installed product** in the window **Installation options**. Select **Continue**.
5. Select **Update** in the **Update** window if you want to update the installed product. Otherwise push **Cancel**.
   Note: The installation program signals if the version that is being used isn't more recent than the one already installed, giving the possibility to continue or stop the installation.
6. Upon completed updating select **Exit**.


Updating from diskettes.

1. Insert the minidisk 1 in unit A.
2. Click the **START** button, then click **Run** from the popup menu.
3. Type A:\SETUP in the **dialog Open field**.

4. Follow the instructions on the screen.
5. Upon completed updating select **Exit**

Updating from CD-ROM.

1. Insert the CD-ROM in the CD-ROM drive. Execute the following procedure (where "**x:**" is the drive letterof your CD-ROM dirve.
2. Click the **START** button, then click **RUN** from the popup menu.
3. Type X:\WINDOWS\INSTENG in the **dialog Open field**.
4. Follow the instructions on the screen.
6. Upon completed updating select **Exit**.

Uninstallation and Reinstallation

Reinstallation of the program is necessary if the user wants to change the installation directory, if problems due to an incorrect installation are manifested, or if the program files have been corrupted or cancelled. To uninstall or reinstall:


Uninstallation Reinstallation from diskettes.

1. Insert the minidisk 1 in unit A
2. At command prompt type A:\INSTALL, press the **Return** key.
3. Follow the instructions on the screen. The program recognizes that a version of BlueCAD has already been installed. Select the option **Delete the installed product and re-install** in the window **Installation options**. Select **Continue**.
4. To uninstall BlueCAD select **Delete** in the **Delete** window.
5. Upon complete uninstallation select **OK** from the **Installation and Maintenance** window.
6. The **Installation** window appears at this point. You can choose to continue the installation, following the installation procedure already illustrated and therefore reinstalling BlueCAD, or to cancel the installation, in this way uninstalling BlueCAD.


Uninstallation Reinstallation from CD-ROM.

1. Insert the CD-ROM in the CD-ROM drive. Execute the following procedure (where "**x:**" is the drive letter of your CD-ROM dirve).
2. At an OS/2 command prompt type **x:** and then press **ENTER** to switch to the CD-ROM drive.
3. Type X:\OS2\INSTENG and press **ENTER.**
4. Follow the instructions on the screen. The program recognizes that a version of BlueCAD has already been installed and proposes the option **Update the currently installed product** in the window **Installation options**. Select **Continue**.
5. To uninstall BlueCAD select **Delete** in the **Delete** window.
6. Upon complete uninstallation select **OK** from the **Installation and Maintenance** window.

7. The **Installation** window appears at this point. You can choose to continue the installation, following the installation procedure already illustrated and therefore reinstalling BlueCAD, or to cancel the installation, in this way uninstalling BlueCAD.

Uninstallation Reinstallation from diskettes.

1. Click the **START** button, then select **Setting \Control Panel** from the popup menu.
2. Select **Add/Remove Programs** in the window **Contol Panel**
3. Select BlueCAD in the **Add/Remove Properties** window.
4. To uninstall BlueCAD click the **Add/Remove** pushbutton.
5. Upon complete uninstallation select **OK**.
6. You can choose to Reinstall BlueCAD following the installation procedure already illustrated

Uninstallation Reinstallation from CD-ROM.

1. Click the **START** button, then select **Setting \Control Panel** from the popup menu.
2. Select **Add/Remove Programs** in the window **Contol Panel**
3. Select BlueCAD in the **Add/Remove Properties** window.
4. To uninstall BlueCAD click the **Add/Remove** pushbutton.
5. Upon complete uninstallation select **OK**.
6. You can choose to Reinstall BlueCAD following the installation procedure already illustrated

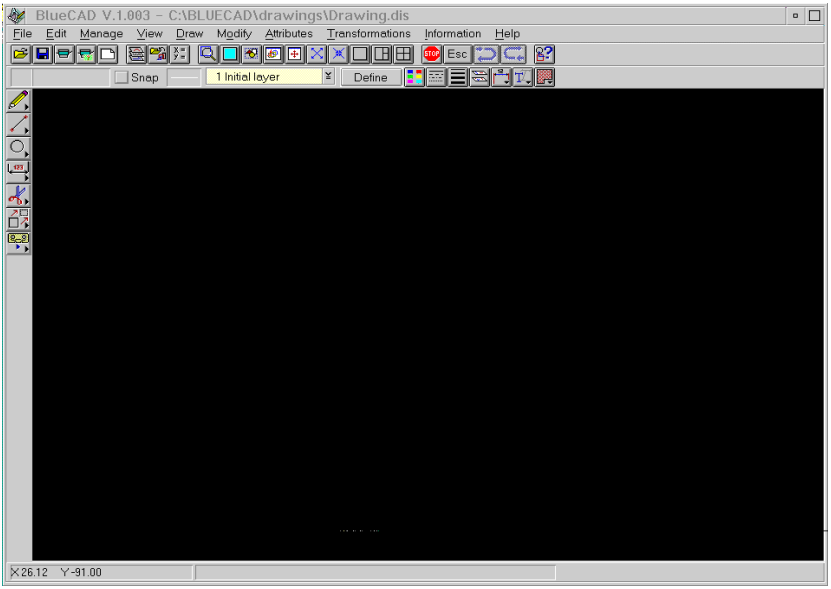
Chapter 2. Let's get to know BlueCAD

In this chapter the BlueCAD elements are described, giving a general look at the work area supplied by the program. The user will know the graphical interface elements and the main modes of interaction with the program are also illustrated.

Starting BlueCAD

To start BlueCAD click twice on the BlueCAD folder

then click twice on the BlueCAD icon contained in it. At this point the BlueCAD window appears. The name of the current drawing, which appears on the bar above the window, is DRAWING.DIS.



The BlueCAD window

The main BlueCAD window consists of the following parts:

Title Bar

Situated along the top part of the window, it contains the name of the program and of the current drawing. The drawing loaded at the starting of BlueCAD is DRAWING.DIS. To move the window:

1. Place the cursor on the **Title Bar.**
2. Hold down mouse button 1.
3. Drag the **Title Bar.**

Note: This procedure also applies to every other BlueCAD *dialog window*

Control Menu

Situated at the extreme left of the **Title Bar**. Click on the **Control Menu** or press **ALT+SPACE BAR** to view the commands that control the dimensions of the BlueCAD window.

Note: A control menu with the same operative modes described now is present in every dialog window.

Minimize to Icon Button

Situated at the extreme right of the **Title Bar**. Clicking this button is the same as choosing **Minimize** from the **Control Menu**: the BlueCAD window is reduced to an *icon.*

Once minimized, press **CRTL+ESC** and click twice on BlueCAD on the **Window list** to view the BlueCAD window again.


Click the BlueCAD icon from **Taskbar**.

Maximize Button

Situated at the extreme right of the **Title Bar**. When you click the **Maximize Button**, the BlueCAD window is enlarged to fill the screen. The window returns to its previous dimensions when you click again.

Menu Bar

Situated under the **Title Bar** along the top part of the window, it contains all the BlueCAD commands. To view the commands belonging to a menu, carry out one of the following operations:

- Click on the menu name.
- Press **ALT+***n*, where *n* is the underlined letter in the menu name. For example, press **ALT+F** to view the **File** menu.

Horizontal Toolbar

Situated along the top part of the BlueCAD window under the **Menu Bar**, its icons show the most useful management and viewing commands to enable quick access.

Upper Status Bar

Situated under the **Horizontal Toolbar**, it contains an information zone, a command echo zone and also an **Attributes Zone** to define and modify the attributes.

Vertical Toolbar

Situated vertically along the left part of the BlueCAD window, it contains the buttons relative to logical groupings of drawing tools (primitive creation tools, entity modification tools, etc.). The user obtain access to the various tool windows through the **Vertical Toolbar**.

For example, to access the **Creation Window**, select the **Creation** button. You will note that the **Creation Window** now appears in a different way to indicate that the **Creation Window** is open.

To close the **Creation Window** use one of the following methods:

- Select the **Creation** button again.
- Select **Close** from the **Creation Window** control menu.

Lower Status Bar

Situated on the lower side of the program window, it is a help zone that serves to view the current coordinates and also the messages sent by the program to the user. In this area a brief description of the current command also appears.

Drawing Area

The remaining area is occupied by the drawing area or the graphic area, which represents the "sheet" on which you draw. When you move the cursor

to the inside of the drawing area, the coordinates appearing in the **Lower Status Bar** identify the current position of the cursor.

Understanding BlueCAD

Getting help

Knowing how to use the on-line information is one of the most important things for a rapid familiarization with BlueCAD. Help information is available for the majority of the BlueCAD graphic interface objects, for each command or menu item and everytime a **HELP** button is present in a program window. BlueCAD makes this information available in various forms.

Help Use: Help Use describes how to approach and consult on-line help. To open this guide carry out the following steps:

1. Select **Help** from the **Menu Bar**
2. Select **Help Use** from the **Help** menu
3. To close the guide select **Exit** from the **Services** menu or press **ALT+F4**, or dubble click on **Close icon button** in the **Title Bar.**

On-line Reference: The on-line *Reference Guide* is the means for quickly finding information while you are using BlueCAD. It provides an explanation of the commands, a detailed description of the graphic interface elements and also an illustration of the fundamental BlueCAD procedures. There are different methods to access the information contained in the *Reference Guide*.

- **Index**
  The index of the guide enables the user to approach the information by subject contained in the *Reference Guide*. To access the index:

  1. Select **Help** from the **Menu Bar**

  2. Select **Index** from the **Help** menu.

3. The index by subject appears from the *Reference Guide* items. To view an index item, click twice on that item. If you wish instead to consult the analytical index of the guide, select the **Analytical index** pushbutton.

4. To close the index select **Exit** from the **Services** menu or press **ALT+F4**.

- **F1 contextual help**
  F1 help is a contextual help, relative to the current BlueCAD command. It is obtained by pressing the F1 key on the keyboard. By doing this you avoid searching for the desired information in the *Reference Guide*. For example, if you wish to get F1 help relative to the **Panoramic** command:

1. Select the **Panoramic** command from the **Vertical Toolbar**, then press the **F1** key. At this point a description of the command contained in the *Reference Guide* will appear.

2. To close the guide window select the **Exit** item from the **Services** menu or push **ALT+F4**, or dubble click on **Close icon button** in the **Title Bar.**

- **Help Button**
  To get help relative to a dialog window you can use, where present, the **Help** button from the same window. As in the case of F1 help, the **Help** button enables the consultation of information relative to a tool at the same time of its use.
  For example, to get information on the **Printer Setup Window**:

1. Select the **File** menu.

2. Select the **Printer Setup** item.

3. Select the **Help** button.

4. To close the guide window select **Exit** from the **Services** menu or press **ALT+F4**.

Note: To learn to use the on-line guide we recommend reading the introduction to the guide (accessible as the *Index* first item). In addition we recommend referring to the **Help** menu of the on-line guide if you would like more information on the consultation methods, including the consultation commands and options (**Search**, **Index**, **Print**, etc.).

- **Fast Help**
  Fast help visualizes a brief description of the command in the **Lower Status Bar**. For example, if you want to get fast help relative to the ellipsis creation command.
  From the menu:
  1. Select the **Drawing** menu.

  2. Highlight the **Ellipsis** command.

  3. This description appears on the lower status bar: `(1)ellipsis given the ends of an axis and another point .`

  From the **Vertical Toolbar**:

  1. Select the **Circle** tool.

  2. Highlight the **Ellipsis** icon.

  3. This description appears on the **Lower Status Bar**:
     (1) `ellipsis given the ends of an axis and another point .`

Note: The procedure described above is valid also if you work in the **Horizontal Toolbar.**

Fast help remains active until one of these events occurs:

- The cursor is placed on the drawing area.
- A new command is selected or highlighted. By doing this the fast help of the new command will appear.

Understanding the commands

Active command and the command echo

The command active at a certain instant is the selected command for which BlueCAD is requiring an input to be able to carry out its execution. The left zone of the **Upper Status Bar** is used to view the echo of the active command, that is the icon relative to the active command and the type of input requested. In the message zone of the **Lower Status Bar** an explanation of the same input appears. For further details and examples see the paragraph *Commands classification* in this chapter.

How to access the commands

The commands can be accessed through different procedures:

- Selecting the relative menu item with the mouse or keyboard. All of the BlueCAD commands are present in the **Menu Bar** and can therefore accessed by this procedure, except for the **Cancel command** and **Data End** command, which are present only in the **Horizontal Toolbar**.

- Selecting the relative icon with the mouse. This is the method to access commands located in the **Horizontal Toolbar** and the **Upper Status Bar**.
- Through the fast selection from the keyboard (short cut), available for the commands used most frequently. The combination of keys for the fast selection of the command can be found, when available, next to its own menu item. For example you can execute the **Print** command by selecting it from the **File** menu or by using the short cut **Ctrl+P**.
- Through the use of the contextual windows. This is the method to access the subcommands, which are the commands logically correlated to the active command. This procedure lets you directly access all subcommands compatible with the current logic context without having to search for them. This allows you to greatly facilitate the drawing process and to learn of the correct use of the work tools. See the following paragraph for an explanation of the subcommands and the access to the contextual windows.
- Through the REXX procedure. This subject will not be dealt with until *Chapter 10*.

The use of one access method as opposed to another obviously depends on the work context. Nevertheless, for fast access to commands it is generally preferable to use the command icons and the function keys rather than resorting systematically to the menus.

Commands classification

The drawing operations have a heirarchy that reflects their use in the drawing process. BlueCAD commands are thus classified in three operative categories, to reflect this logic and speed up the drawing phases:

- **Main commands**
  Main commands are commands that can be executed in any moment and they immediately remove any other previously active command from the BlueCAD input. A main command usually remains active upon completed execution, therefore enabling its subsequent rerun. The main commands are marked by the number (1), appearing, together with a brief description of the command, when you activate fast help. For example, all the commands from the **Draw** and **Modify** menus and most of the commands from the **Vertical Toolbar** are main commands.

1. Select the **Segment** button from the **Vertical Toolbar**.

2. Select the **Segment 2 Points** icon from the **Segments Window**. Note the command echo: the command icon and the type of input required, `Point`, are visualized in the **Upper Status Bar**.

3. Now supply the first extreme of the segment by selecting a point of the graphic area.

4. Now supply the second extreme of the segment by selecting a point on the graphic area. The execution of the command is now complete and the drawn segment is present. The command **Segment 2 Points** is still active and BlueCAD is ready for its rerun: in fact, you note that the command icon and the type of input required, `Point`, are not changed in the **Upper Status Bar**.

5. Draw a second segment by repeating steps 3 and 4.

- **Transparent commands**
  Transparent commands are commands that can be executed in any moment, without interfering with the current state of the BlueCAD input. Transparent commands are automatically disactivated after execution. Transparent commands are symbolically marked by the

number (0). This number appears, together with a one-line description of the command, activating the Fast Help relative to the command. Among others, transparent commands are all the commands from the **View** menu.

1. Select the **Positive Zoom** icon from the **Horizontal Toolbar**, that brings about an enlargement of the visualized graphic primitives with a ratio of 2:1.

2. Since it is a transparent command, it has not interfered with the previous input status upon its execution. In fact, the previous command, **Segment 2 Points**, is highlighted by the eco command: in the Upper Status Bar, the icon of the **Segment 2 Points** command is still active and the type of input requested, `Point`, has not been modified.

- **Subcommands or secondary commands**
  Subcommands or secondary commands are commands that can be carried out only if a main command compatible with it is already active. The execution of a subcommand outside of this condition is signaled by BlueCAD in the **Lower Status Bar** with the message: `Command not compatible`. Access relative to subcommands is through the contextual windows, activated by a click on the graphic area with mouse button 2.

  Important: There are two main contextual windows exist: the **Snaps Point Window**, that can be activated each time BlueCAD requests a point as input, and the **Selection Window**, that can be activated each time BlueCAD requests the selection of a primitive as input. Therefore remember that each time a point type or selection type input is requested you have all the subcommands from the corresponding contextual window at your disposal by clicking mouse button 2.

  The subcommands leave the current state of the BlueCAD input unaltered, by not removing the previously active command. The subcommands are symbolically marked by the number (2). This number appears, together with a one-line description of the command, when you activate fast help.

1. Select the **Modify** button from the **Vertical Toolbar**.

2. Select the **Delete** icon from the **Modify Window**. The **Delete** command is a main command, therefore it removes the previous **Segment 2 Points** command from the input. Note that the **Delete** command icon is now present in the echo zone and that the type of input required is `Selection`, that is you must select the primitives you wish to delete.

3. Click mouse button 2 in the graphic area. The contextual **Selection window** appears, which shows the selection subcommands available.

4. Select the subcommand **Select all** from the contextual menu to delete all the drawn primitives.

5. Note how the subcommand hasn't interfered with the status of the previous input during its execution. In fact, the previous **Delete** command is still active, as made evident by the echo command: the **Delete** command icon still appears in the **Upper Status Bar** and the type of input required, `Selection`, has not been varied.

Note: Any command can be removed, in this way resetting the BlueCAD input status, executing any other main command, or selecting the **Cancel Command** command.

Exiting from BlueCAD

It is possible to exit from BlueCAD using one of the following methods:

- Select the **Exit** command from the **File** menu.
- Use the short cut **Ctrl+X**.
- Click twice on the **Control Menu**
- Select the **Close** command from the **Control Menu**
- Use **Close icon button** in the **Title Bar.**
- Use the short cut **Alt+F4**.

Note: Since drawing operations have been made in the previous paragraphs, BlueCAD will warn that the drawing has been modified and it will ask you to save the changes: click the **NO** pushbutton if you don't want to save them or **YES** if you do. Click **Cancel** to cancel the exit procedure.

Chapter 3. Basic operations

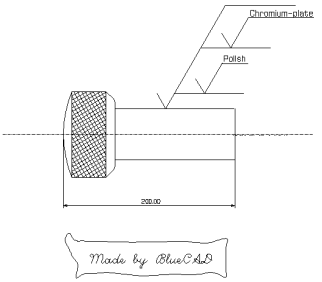
At this point you should have a certain familiarity with the main BlueCAD window, you should know how to access on-line help information and you should also have understood how the commands are organized and how to access them. It is important that these subjects have been fully assimilated before proceeding further in reading this chapter. In this case you can begin to become familiar with the basic BlueCAD operations.

These operations are illustrated by means of an example drawing that the reader is called to execute during the reading. At the end of the chapter these notions necessary for drawing with BlueCAD will be taken and put into practice. Many of the subjects treated will be taken up again, in more detail, in the next chapters.

The drawing procedure

BlueCAD's basic work procedure consists of the following steps:

- Opening a new drawing or an already existing drawing
- Drawing using the BlueCAD tools
- Saving the drawing
- Printing the same drawing



In the course of this chapter these aspects are considered through the execution of a simple example drawing. At the end of this chapter, the drawing in the figure will therefore be drawn, saved and printed

You can now start BlueCAD.

Opening a new drawing

The **File** menu contains the commands to open, save and rename drawings.

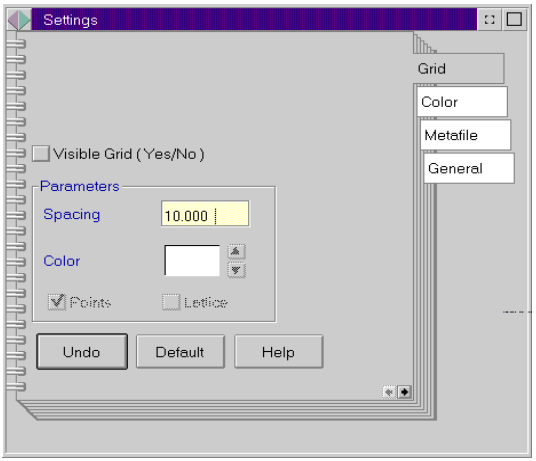
Follow these procedures to open a new drawing:

1. Select the **File** menu.
2. Select the **New** command.
3. Select the **YES** pushbutton upon request for confirmation.

Note: you can open a new design also by selecting the **Open** icon from the **Horizontal Toolbar**.

Use of the Grid and Snap tools

Once BlueCAD has been started and a new drawing opened, you can change the BlueCAD settings to use the*Grid* tool. The Grid can be compared to a multimetered paper, showing a reference grid in the drawing area. Such a grid is not part of the drawing and is therefore not printed. The Grid is of great help to the drawing because it facilitates the creation and the positioning of the graphic entities, enabling the use of relative references rather than absolute coordinates. To set up the Grid:



1. Select the **File** menu.
2. Select the **Settings** command. In this way you open the**Settings Window**. This window looks like a notebook composed of many pages. Each of the pages is individuated by a bookmark with its name. The **Settings Window** is already opened to the**Grid** page.
3. Select the bistable**Visible Grid** button. The *check mark* on the button indicates the selection.

4. Close the **Settings Window** by clicking twice on the control menu. At this point a grid having step 10 is represented in the graphic area.

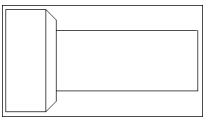
The *Snap* tool, which doesn't have an equivalent in the manual drawing, serves to force the cursor movements in the graphic area, so that the cursor is "magnetized" by the *snap points*. By doing this you check the movement of the cursor with precision and the geometric constructions are facilitated, allowing snap points to be used as reference points. You can restrict the use of the cursor to different types of snap points, specifying both Grid points and points belonging to primitives (end points of primitives, mid point of primitives and others).

Now you want to use the Snap tool in conjunction with the Grid, therefore activating Snap and selecting the Grid points as snap points:

1. Select the **Snap** command from the **Horizontal Toolbar**. The **Snap Contextual Window** will appear, which indicates the different types of snap points that can be activated.
2. Select the **Point on grid** command. The contextual window closes automatically. The check mark on the **Snap** button will appear to indicate that the automatic snap is now active.
3. Observe that the cursor has changed it appearance, this signals that it is in snap mode; also note the presence of a green echo of the cursor. Now move the cursor in the drawing area: while the cursor has a continuous movement like always, its echo has a discrete movement, anchoring itself to the nearest grid point; this is the point that is selected if you click on a point in the graphic area.

Thorough explanations of the Snap and Grid tools are provided in the next chapter. Now it is time to draw.

Drawing segments



At the end of this paragraph the drawing in data processing will appear as in the figure.

Let's start by drawing the rectangular head of the bolt which has the lower left vertex on the point (-100, -70) and the upper right vertex at (-60, 30):

1. Select the **Segments** button from the **Vertical Toolbar**: the **Segments window** will appear.

2. Select the **Rectangle** command from the **Segments window**. The echo command signals that the input requested is a point, corresponding to the lower left vertex of the rectangle.

3. Select the *coordinates* (-100, -70) using one of the two following ways:

- Select the **Coordinate** button from the **Horizontal Toolbar**. The **Coordinates Window** appears; this window enables the insertion of a point, specifying its coordinates: in the **X=** field you enter -100, in the **Y=** field you enter the value -70. Select **YES** or press the **Return** key to confirm the entered coordinates.

  Note: The **Coordinate** command can be carried out by the **Manage** menu or from the **Horizontal Toolbar**.

- Move the cursor on the grid until the coordinates (-100, -70) are indicated in the **Lower Status Bar**. This point is a point on the grid (and precisely 16 grid steps on the right and 14 on top in respect to the lower left end point), therefore, by the snap being activated on the grid, the discreet movement of the cursor enables a rapid individualization. Click on the point individuated in this way.

4. The echo command signals that the input requested is still a point, corresponding now to the upper right vertex of the rectangle. Select such a point, with coordinates (-60, 30), using one of the two following methods:
   - Enter the coordinates -60 and 30 in the **X=** and **Y=** fields of the **Coordinates Window**
   - Move the cursor in the graphic area until the echo coordinate of the **Lower Status Bar** shows the desired coordinates (-60,30). The grid step being set at 10, you find such a point at 10 grid steps up and 4 to the right in respect to the first vertex of the rectangle.

The **Coordinates Window** can be automatically activated each time a point is requested as input. It enables you to specify the coordinates instead of clicking on a point in the graphic area. There are some aspects to underline concerning the operative methods of the **Coordinates Window**

- The **Coordinate** command is a bistable (two-state) button, that is of the ON/OFF type: this means that by carrying out the command, the **Coordinates Window** changes status, becoming active (ON status) if previously unactive (OFF status) and viceversa. By activating the **Coordinates Window**, each time BlueCAD requests a point type input, it is automatically appears, giving the user the possibility to specify the point through its coordinates.
- The coordinates present when the **Coordinates Window** is opened, are those of the last selected point. In this way it is possible to specify the coordinates of the point in relation to the last input.
- The coordinates can be entered in the **Coordinates Window** also by means of expressions that use the functions and syntaxes of the **Scientific Calculator** integrated with BlueCAD. For further details regarding these functions see the on-line *Reference Guide.*

  Note: The possibility of specifying a numerical input through the use of expressions and functions applies not only to the **Coordinates Window** but each time a numerical type input is requested. For this reason it will not be explicitly mentioned again.

Let's now complete the first part of the drawing:

1. Select the **Segment Piecewise Linear** command from the **Segments Window**. The echo command asks for a point type input again. The **Segment Piecewise Linear** command is useful for drawing consecutive segments.
2. Click successively: on the upper right vertex point of the already drawn rectangle (coordinate point (-60,30)), one grid step on the bottom and one on the right (point (-50,20)), six grid steps on the bottom (point (-50,-60)), one on the bottom and one on the left ((-60,-70)).
3. Select again the **Segment Piecewise Linear** command.
4. Click successively on the points, refering to the last point previously selected: one grid step on the right and two on the top ((-50,-50)), fourteen on the right ((90, -50)), six on the top ((90, 10)), fourteen on the left ((-50,10)).

Note: If the Grid tool wasn't used, you could use the **Horizontal-Vertical** command from the **Segments Window** to trace horizontal and vertical segments. The presence of the Grid has made the use of this command superfluous, as the use of the **Coordinates Window** to specify the coordinates of the selected points.

BlueCAD offers many commands to draw segments, other than obviously the **2 Points** command (ends given segment). In this way, many geometrical constructions, otherwise laborious, turn out immediately.

If you make a mistake: Undo, Repeat and Cancel Command

It is possible to immediately undo the effects of an undesired command with the **Undo** command. You can repeatedly carry out the **Undo** command in such a way to undo the executed operations starting from the last, until there aren't anymore operations to undo or until you have reached the maximum number of 100 undone operations.

Use the **Repeat** command if you want to recuperate an undone operation; the operations are recuperated starting from the last.

The **Undo** and **Repeat** commands are accessible from the **Edit** menu, and also from the **Horizontal Toolbar**.

1. Select the **Undo** button from the **Horizontal Toolbar**: the last executed operation is undone and therefore the segment of end points (50, 10) (90,10) is cancelled..
2. Carried out repeatedly the **Undo** command: the executed operations will be undone in sequence; at the end a beep and the disqualification of the command icon signal that there aren't anymore operations to undo. At this point you will have returned to the drawing beginning, without any drawn primitive.
3. Select the **Repeat** button from the **Horizontal Toolbar**: the rectangle of the head of the bolt reappears, thus recuperating the last undone operation.
4. Execute the **Repeat** command again until the executed drawing is restored. The fact that there aren't anymore operations to restore is signaled by a beep and the disqualification of the command icon.

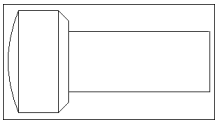
If you wish to remove an active command, thus resetting the input status of BlueCAD, you can do it by selecting the **Cancel Command** command from the **Upper Status Bar**, through the **Esc** key on the keyboard, aside from selecting any main command. This is useful, for example, when you enter a wrong input value while carrying out a command.

The redraw operation

The redraw operation serves for recleaning and updating the drawing area when the visualization of the drawing seems to be missing some parts after the cancelation and modification operations described in the previous paragraph. This operation, like any other visualization operation, is transparent and can therefore be used at all times, also during the execution of other commands, without changing the input status.

1. Click the **Redraw** button from the **Horizontal Toolbar**. In this way the visualization of the drawing is "renewed", making unviewed parts newly visible.

Drawing an arc

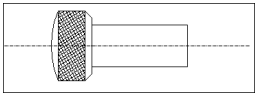


Now you will draw the *arc* of the bolt head, in such a way that the drawing results as in the figure:

1. Open the **Circles Window** by selecting the **Circle** button from the **Vertical Toolbar**.
2. Execute the **Arc** command, which requests three points belonging to the arc.
3. Click on the lower left vertex of the rectangle of the bolt head, the first end point of the arc.
4. Select a point of the arc by clicking one grid step on the left and five grid steps at the top of the grid.
5. Click on the upper left vertex of the rectangle of the bolt head, the second end point of the arc.
6. Close the **Circles Window** by selecting the **Circle** button again.

Definition and modification of attributes

Now let’s draw the hatching of the bolt head, by hatching the closed rectangular perimeter of the head.



Let's also draw the line of symmetry, in a way to obtain the above drawing. Therefore, let's pause briefly on the *attributes* of the graphic primitives and on the way in which you can operate on.

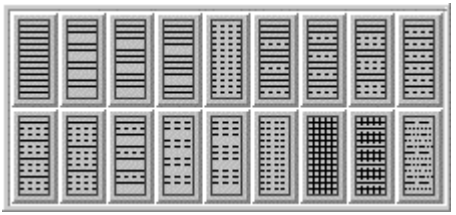
The commands that operate on the attributes are indifferently accessible from the **Attributes** menu and the **Attributes Zone** in the **Upper Status Bar**. There are the commands relative to the general attributes (color, thickness, line type and layer), of every type of graphic primitive, and the commands relative to the specific attributes of the dimension primitives, texts and hatchings. In the case of hatchings, for example, the specific attributes are the type, *slant* and the step of the hatching.

There are two ways to operate on the attributes of the graphic primitives in BlueCAD: the definition mode and the modify attributes mode. These two modes are alternatives and you can activate one or the other with the **Define** and **Modify** commands in the **Attributes** menu , or equally with the two-state **Define-Modify** button in the **Upper Status Bar**. The values of the attributes, set when the define attributes mode is active, remain current until you substitute them with different values. This means that these automatically become the attributes applied to the primitives that they will create. Vice versa, with the modify mode, it is possible to modify the values of the attributes of the primitives already created without modifying the current values of the attributes and therefore without influencing the attributes of the primitives that will be created next.

Let's now draw the hatching by applying what we have learned so far.

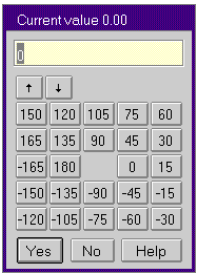
1. Activate the define mode, if it isn't already active, by selecting the **Define-Modify** button.
2. Select the **Hatching Attributes** button from the **Upper Staus Bar**. This opens the **Hatching Attributes Window**.

3. Select the **Hatching Type** command, which opens the **Hatching Type Window**.



4. Set the desired hatching type by selecting the third to the last hatching type in the lower line of the **Hatching Type Window**. The **Hatching Type Window** automatically closes.
5. Repeat the operation from number 2.

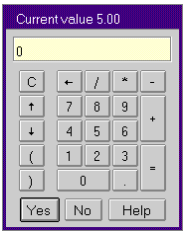
6. Select the **Hatching Angle** command, which opens the input **Angles Window**.
7. Enter the desired angle, 30, of the hatching slant in the **Angles Window**. This can be done by either typing 30 in the entry field or by selecting the 30 button on the little keyboard in the **Angles Window**. There are buttons relative to the most commonly used angles on this keyboard to speed up the input operation.



Note: the **Angles Window** is the window that is always used when an angle type input is requested. See the on-line *Reference Guide* for details on the operative methods of this input window. We stress that the angles are always specified in degrees and that an angle is positive for a counterclockwise rotation and negative for a clockwise rotation.

8. Select **YES** or press **Return** to confirm the entered value and to close the **Angles Window**.
9. Repeat the operation from number 2.

10. Select the **Hatching Step** command, which opens the **Normal Calculator Window**.
11. Set a step (spacing between two consecutive hatchings) equal to 4 in the **Caculator Window**. The input can be provided by either typing the desired value in the entry field or by using the little keyboard in the **Calculator Window**.



Note: the **Calculator Window** is the window that is always used if a numerical type input is requested. For details on the operative methods and for a more advanced use of this window, see the on-line *Reference Guide*.

12. Select the **YES** pushbutton or press the **Return** key to confirm the entered value and to close the **Calculator Window**.

The attributes (type, angle, step) of the hatching are defined in this way. The hatching that will be drawn will therefore inherit these attributes.

1. Select the **Creation** button, which opens the **Creation Window**.

2. Select the **Hatching** command. The cursor, now having the selection trap form, and the command echo `Selection,` indicate that a selection is requested to execute the **Hatching** command: you need to simply select only one primitive composing the closed perimeter that you want to hatch, as much the individualization of the remaining primitives component the perimeter comes automatically.
3. Select any side of the bolt head: the hatching according to the previously defined attributes is drawn.

   Note: Since the two longer sides of the bolt head also belong to other closed perimeters when you select one of them; to avoid the misunderstanding of which is the hatching area, make sure that the center of the selection trap is inside the perimeter that you want to hatch (in this case, the head of the bolt).

Let's now draw the symmetry line:

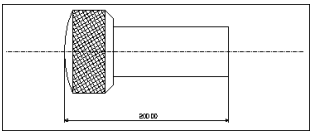
1. Select the **Segment** button to open the **Segments Window**.
2. Select the **Horizontal-Vertical** command.
3. Click, with reference to the lower left vertex of the bolt head, 10 grid steps on the left and 5 on top ((-150,-20)), and 10 grid steps on the right and 6 on top in respect to the lower right vertex of the bolt stem (140, -20). In this way the symmetry line is drawn.
4. Now modify the line type continuously to the point type. Activate the modify attributes mode by clicking on the **Define-Modify** button.

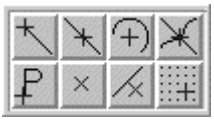
5. Select the **Line Type** button to open the **Line Type Window**.
6. Set the desired line type by selecting the fourth line type from the **Line Type Window**. The **Line Type Window** automatically recloses. The cursor now has the shape of the selection trap, indicating that the selection of the primitives, on which to apply the set modification, is requested.
7. Click on the symmetry line, modifying in this way the line type continuously to the point type.

Note: As a logical consequence of what you have already learned in the previous chapter, you note how all the attribute commands (for example, those just used like **Hatching Type, Hatching Step, Hatching Angle, Line Type**) are transparent commands if imparted when the define attribute mode is active, while they are main commands if you activate the modify attribute mode.

Drawing a dimension

To have complete information on a drawn object you must know its dimensions: these are provided by the *dimensions* entities. We now draw the dimension relative to the length of the bolt to obtain the drawing in the figure:





1. Open the **Dimensions window** by selecting the **Dimensions** button from the **Vertical Toolbar**.
2. Select the **Dimension 2 Points** command, which requests as input the two points from which the measurement is taken and the positioning point of the dimension text. In our case, the two points to value are the mid point of the arc of the bolt head and the mid point of the right side of the stem.
3. To select the mid point of the arc we will use the snap. Click on button 2 of the mouse to view the contextual **Snap Points Window**

4. Select the **Mid Point** command to temporarily qualify the snap on the mid point of the primitives.
5. Click on any point of the arc in question: since the snap is activated on the mid point, its mid point selected.
6. Repeat the operations from points 3 and 4.
7. Click on any point of the side in question: its mid point is automatically selected.
8. Moving the cursor on the graphic area, a *creation echo* shows the preview of the dimension, that is how the dimension will be created. Click on the point in the graphic area in which you want the dimension text to be positioned. In this way the dimension is created.

Note: The points to value could also be individuated by their belonging to the grid. The Snap and the **Snap Points Window** have been used to familiarize you with tools that will become habitual.

Saving a drawing

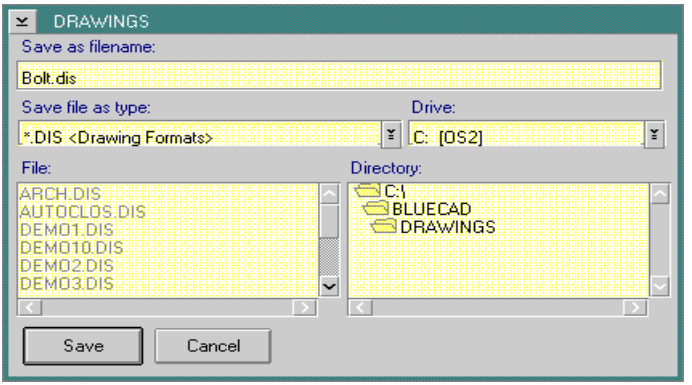
Saving a drawing is necessary for memorizing it in a *file* on the *disk*: if you exit from BlueCAD without carrying out this operation, the work done since you last saved it will be lost.

For this reason it is a good rule to remember to save periodically while you work. Saving protects your work against unforeseen events and against malfunctions of the PC and of the electric feeding.

When saving, a name is assigned to the drawing file. When a new drawing is opened, it is always created with the default name of DRAWING.DIS. For this reason, when a drawing is saved for the first time, you provide it with a unique name that can identify it.

To save the drawing, giving it the name of BOLT:

1. Select the **Save as** command from the **File** menu. This opens the **Save as Window**.



2. You now have the possibility the change the assigned disk, directory and file extension values.
3. Type `Bolt` in the entry field.

   Note: We recommend that you choose the drawing names in a way that they recall the content, for faster identification.

4. Select the **Save** pushbutton to confirm the typed text. The drawing is saved in the specified disk and directory with the name BOLT.

   Note: In case the name of a preexisting drawing is attributed to the drawing (for example if in this case a file named BOLT.DIS already existed), saving implies the overwriting of the drawing. To avoid the consequences of wrong behavior, BlueCAD notifies this occurence by asking confirmation when saving.

The **Save as** command can be used successively if you want to change the name of a drawing; to save the drawing in the future, execute instead the **Save** command from the **Horizontal Toolbar** or from the **File** menu, which writes the new version over the old. The old version is conserved in a safety copy having the same name and file extension .BAK.

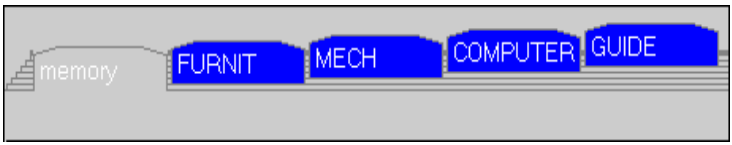
BlueCAD provided you with an additional safety system that guarantees the integrity of the drawing and the recovering of the work done in the case of unforeseen malfunction by the system. In fact, BlueCAD provides an automatic saving of the drawing with the name AUTOCLOSE.DIS, freezing the drawing at the status preceding the malfunction.

Using a block

More primitives can be grouped into one primitive, creating in this way a *block*. The blocks can be saved in *libraries of blocks*, from which they can subsequently withdrawn to be used in the current drawing. Now you want to position in the drawing a block representing a working:

1. Click on the **Manage Blocks** button or select the **Blocks** command from the **Manage** menu to open the **Manage Block Window**. This window has the aspect of a notebook composed of many pages. Each page represents a library of blocks and is individuated by a bookmark having the name of the library itself.
2. Press mouse button 2 when the cursor is in the zone of the bookmarks. By doing this a contextual window for the creation and cancellation of the blocks library appears: select **New Blocks Library**. The **Create New Library Window** appears. Type `GUIDE` in the entry field, then press **YES** or **Return.** The library is joined to the **Manage Block Window**, together with the corresponding bookmark.

3. Select the **GUIDE** bookmark to open the Guide library containing the block to be positioned. Note how each block is identified by means of its name and by an icon that reproduces the block itself.



4. Drag and drop the WORKING block in the graphic area. The command echo signals that a point is requested as input; such a point is the positioning point of the block. Note how a dialog window, the **Block Positioning Window**, is automatically opened; this window enables you to check the scale and rotation angle of the block. A creation echo of the block, visible by positioning the cursor in the graphic area, provides a preview of the block.
5. Choose a point belonging to the upper side of the stem as a positioning point by clicking on it.
6. If you want to close the **Manage Block Window**, select the **Manage Blocks** button again or the **Blocks** command from the **Manage** menu, or click twice on the **Control Menu**

Making an enlargement

While working it is often necessary to modify the view of the drawing, obtaining a view more or less in detail of its particulars. To do this, use the the viewing commands accessible through the **View** menu or from the **Horizontal Toolbar**. To view in detail the bolt head:

1. Select the **Enlargement** command from the **Horizontal Toolbar** or from the **View** menu. The command echo signals that a selection is requested, while the aspect of the cursor also changes, indicating that a zone to be enlarged must be selected.
2. The rectangular zone to be enlarged is specified by means of the lower left vertex and its upper right one. Therefore click on a point in the graphic area which is slightly under and to the left with respect to the bolt head.
3. The echo command shows a preview of the zone that will be enlarged. Move the cursor in a way that such a zone completely contains the bolt head and click. The selected zone will be enlarged, occupying the entire graphic area.

Note: Since the **Enlargement** command is a transparent command, it does not influence the previous input status, therefore the **Block Positioning Window** remains open.

Note: When you carry out the **Enlargement** command, like every other command from the **View** menu, a redrawing operation is also automatically carried out, thus updating the graphic area.

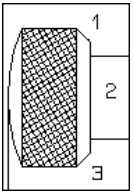
Modifying a primitive

Now you want to*fillet* the segments that make up the insertion base of the stem.

1. Click on the **Modify** button from the **Horizontal Toolbar**.
2. Click on the **Fillet** command. The **Calculator Window** opens.

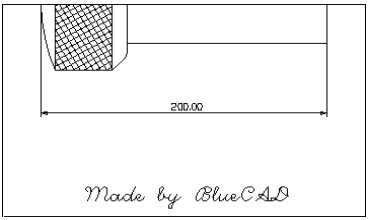
   Note: Since the **Fillet** command is a main command, note that the previous input status is reset and therefore the **Block Positioning window** is automatically closed.

3. Using the little keyboard of the calculator or the keyboard, type the value 6 as fillet radius in the entry field. Select **YES** or press **Return** to confirm the entered value.



4. The echo command indicates that the two primitives to fillet must be selected. Click on the segments 1 and 2.
5. Since the fillet command, is a main command, it remains active, with the radius value. To fillet segments 2 and 3, just click on these segments.

Drawing a text



We now want to draw the `Made by BlueCAD` text that appears in the lower zone of the drawing.

To do this, proceed preliminarily to view the drawing in all of its parts:

1. Select the **Zoom All** button from the **Horizontal Toolbar**. This command enables the viewing of the total drawing in the maximun dimension compatible with the available graphic area. You can also access this command from the **View** menu.

Now let's draw the text:

1. To define the *font* of the text, activate the define attribute mode, if it isn't already active, using the **Define-Modify** button in the **Upper Status Bar**.

2. Select the **Text Attributes** button from the **Upper Status Bar**. The **Text Attributes Window** opens.

3. Select the **Text Font** command. The **Calculator Window** opens, as the identifying number of the font is requested.
4. Type 4 in the entry field of the calculator and click **YES**.
5. Click on the **Creation** button of the **Vertical Toolbar**.

6. Click on the **Text** command. The input **Strings Window** opens.
7. Type the text `Made by BlueCAD` in the entry field of the window. Select **YES** or press **Return** to confirm the entered text.

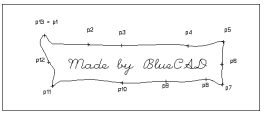
   Note: The **Strings Window** is the window that is always used when a string type input is requested. For more details on the operative methods of this input window see the on-line *Reference Guide*.

8. The creation echo now shows the overall box of the text. The command echo signals that the positioning point is the requested input. Using the **Coordinates Window**, provide the input (-70, -170) and click **YES** or press the **Return** key.

Drawing a spline

Let's now draw the "frame" of the text, using a *spline*, which is a curve specified through a sequence of points which belong to it, called nodes.

1. Click on the **Creation** button from the **Horizontal Toolbar**.
2. Click on the **Spline** command. The sequence of the spline nodes is now requested as input.
3. Select the first node on the point p1(-100, -130).
4. Select the points p2.....p12 in sequence
5. Select the last node, p13 in such a way that it coincides with p1.



6 Select the **Data End** command from the **Upper Status Bar**. This notifies the program that the input has been concluded.

We have completed the example drawing.
Let's now save the drawing:

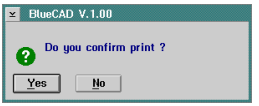
1. Select the **Save** command from the **Horizontal Toolbar** or from the **File** menu.
2. A window with the message `Save and substitute the drawing` appears: click **YES** pushbutton or press the **Return** key.

To terminate this exercise, we can now print the drawing

Printing the drawing

Without entering into printing and plotting details, for now let's limit ourselves to printing the created drawing.

1. Select the **Print** button from the **Vertical Toolbar**. You can also access the **Print** command from the **File** menu.



2. Click on **YES** or press **Return** at the confirmation window, `Confirm print.`
3. A window shows the advancement of the printing by percentage.

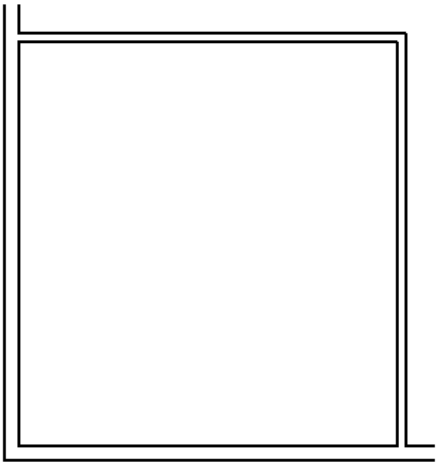
Second Part - Drawing Techniques

In the First Part, the basic operations of BlueCAD have been illustrated by the execution of a simple drawing. In the Second Part you are therefore ready to closely examine BlueCAD's drawing techniques and tools. The drawing of a room plan, which the reader is called to carry out during the reading, will be used for description purposes. The knowledge acquired will be indispensible not only in the architectural drawing, but also in general, to be able to use BlueCAD for any type of drawing.
In case the user is not familiar with BlueCAD, it is a good rule to proceed further only after having read what was explained in the previous chapters. In particular, the user should have acquired a certain familiarity with the BlueCAD window, the commands' classification and their access. The user should also know how to perform the basic drawing operations illustrated in Chapter 3.

Chapter 4. Creating a drawing

This chapter deals with the settings of the work environment. These enable you to prepare the drawing instruments, before creating a drawing.

In this chapter the use of basic drawing techniques, such as the relative positioning, modifying and deleting of primitives, will be introduced. The following drawing, representing the walls of a room, will be created using these techniques.



The project of the room will be used to illustrate BlueCAD's functions until *Chapter 7*.

Settings of the work environment

Before starting to draw we organize settings of the work environment to meet our requirements. Becoming acquainted with the settings options will help you to make better use of BlueCAD, furthermore making important drawing tools available.

The Settings Window

The basic options for personalization are grouped together in BlueCAD's **Settings Window**. In the previous chapter we have already used these in order to qualify the grid. Open the **Settings Window** in order to see and eventually modify the current settings in BlueCAD:

1. Select the **File** menu.
2. Select the **Settings** command to open the **Settings Window**. This window looks like a notebook made up of several pages. Each page is identified by a book marker which bears its name.

To open a page from the **Settings Window** all you have to do is select the relative bookmark. The **Settings Window** contains the following four pages:

- **Grid Page**. It contains the options relative to the Grid, whose significance and usefulness was seen in the previous chapter.
- **Color Page**. The colors that you want to use in the drawing can be chosen from this page.
- **Metafile Page**. On this page you can choose where to copy the drawing or parts of it when you want to export it.
- **General Page**. This is the page for defining the general settings of BlueCAD.

In the lower part of each **Settings Window** there are the three buttons **UNDO**, **DEFAULT**, and **HELP**:

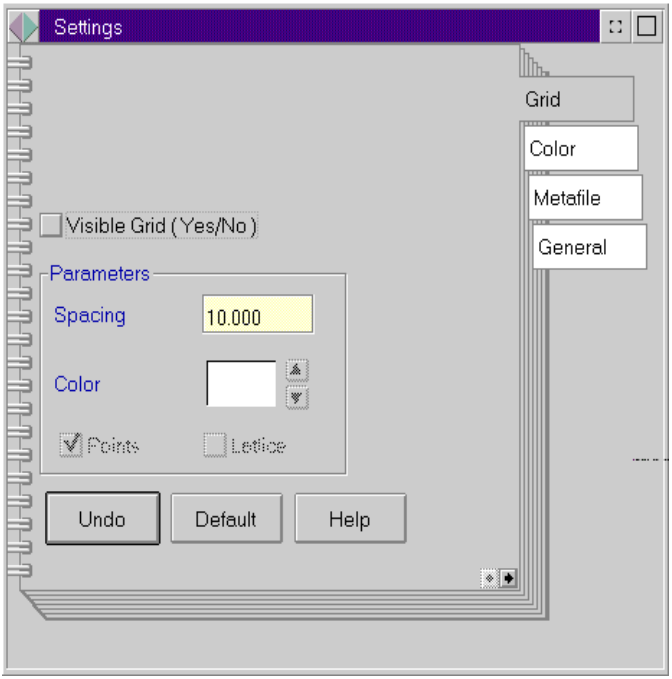
- **UNDO** button: if the settings of the current page are modified, these modifications will be cancelled by selecting **UNDO** to re-establish the previous settings.
- **DEFAULT** button: by selecting **DEFAULT**, the undertaken settings will be assigned to the current page.
- **HELP** button: by selecting **HELP**, the contextual help appears.

Note: The modifications made on the settings will be effective as soon as the **Settings Window** is closed. These settings are memorized by BlueCAD and the undertaken settings remain, even after restarting BlueCAD, until remodified

The **Settings Window** can be closed by a double click on its control menu or by pressing **ALT+F4**.

Setting the Grid

Open the **Grid Page,** if it isn't already open, by selecting the relative bookmark.



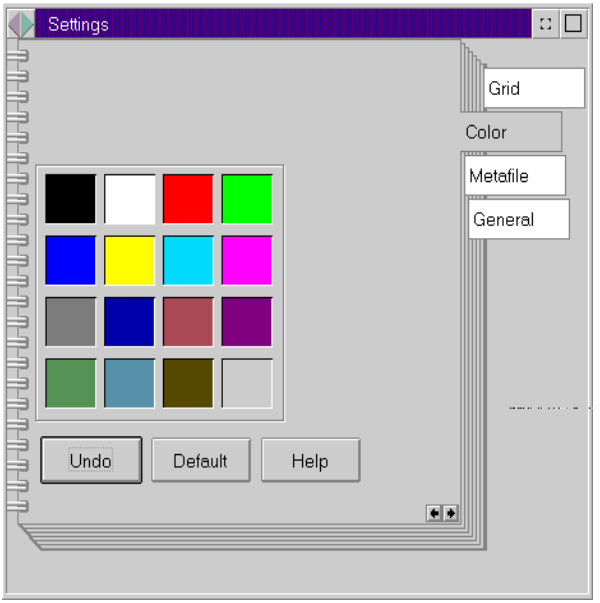
The upper button of the **Visible Grid (Yes/No)** is a two-state button: by selecting it you can activate or deactivate the use of the grid instrument.

The Visible Grid option is active if there is a check mark on the button. Make the grid active.

If you activate the Grid, you can set the following parameters of the Grid on the **Parameters** frame:

- **Step:** The Grid step is the difference between the coordinates of two points next to each other on the Grid; the step determines the density of the Grid nodes. Enter the value 5 into the entry field, to set the step to the value that we start drawing the plan.
- **Color:** using the suitable arrows, you can selecte the color with which to view the Grid.
  Note: Using the same color of the graphic area for the Grid obviously doesn't allow the Grid to be distinguished from the background.
- **Points - Squares**: the two options cannot be used together. Select **Points** if you want to view the Grid with points, select **Squares** if you want to see it with squares.

Setting the colors



Open the **Color Page** by selecting the relative bookmark. On this page the palette of colors used in BlueCAD can be seen. In case you want to use different colors than the ones shown, this palette can be modified by the following procedure:

OS/2 WARP

1. Open the **OS/2 System** folder which is present in your desktop.
2. Open the **System Setup** folder.
3. Double click on the **Color Palette** icon.
4. Drag the color of your choice from the palette and drop it on the color that you wish to modify on the BlueCAD color palette. Repeat the same operation for all the colors that you want to modify.

Note: If the modified colors have been used in the current drawing they will also be automatically substituted in the respective primitives of the drawing.

In case you want to modify the background color of the graphic area, follow the above mentioned procedure, dropping the color on the graphic area instead of on the BlueCAD palette.

Nota: A similar procedure can be followed if you want to modify BlueCAD's menu fonts: carry out steps 1 and 2 and then open the **Font Palette,** dragging and dropping the prechosen font on BlueCAD's **Menu Bar.** This proceedure can only be used for BlueCAD's graphic interface fonts and not for drawing ones.

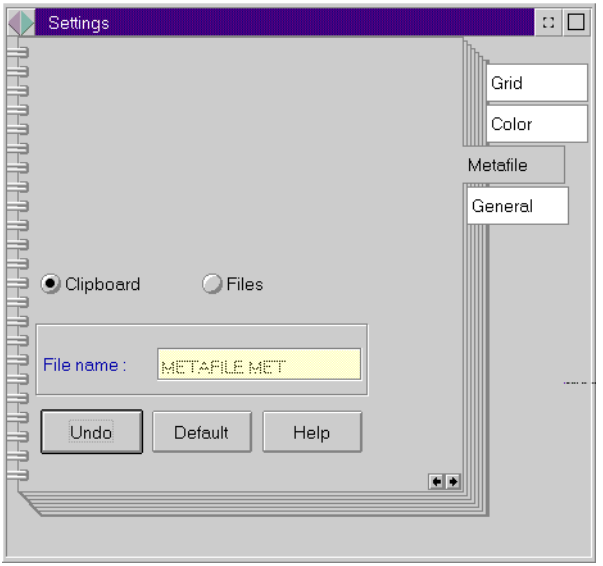
1. Double click on the color to modify in the **Color Window** opens the window **Color**.
2. Select the new color and press **Enter**. In the color window is now present the new color.

It is also possible to to choose custom color:

1. Repeat step 1.
2. Select custom color and press **Define Custom Color** button.
3. Select the color in the Large Color Box
4. Press the **Add to Custom Color** button and press **Enter**.

Metafile setting

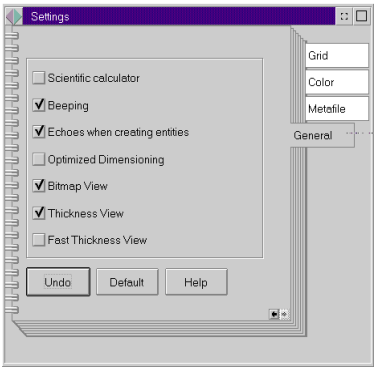
Open the **Metafile Page,** selecting the relative bookmark

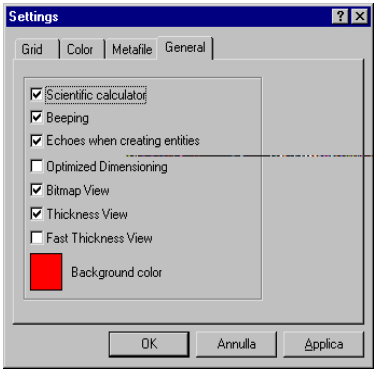


The *metafile* is a format used when you want to copy the drawing or a part of it so that it can be used by other programs.This will be dealt with in *Chapter 10*. For the moment it is sufficient to know that this option exist. ***Clipboard*** and **File**, reciprocally incompatible, enable you to specify where to copy the drawing that you want to transfer either in the clipboard or in the specified file. The default option is**Clipboard**.

General settings

Open the **General Page** selecting the relative bookmark.





The options that are present can either be activated or not by selecting the relative two-state buttons. If an option is activated there will be a check mark on the button. In carrying out our project, the default settings will be considered chosen

- **Scientific calculator:** this option activates the use of the **Scientific Calculator Window**, instead of the **Normal Calculator Window**, for every numerical input. For ordinary use, the **Scientific Calculator Window** may seem to be very complex; usually the **Normal Calculator Window** is faster and more functional, and so it is assumed by default. However, it can sometimes be useful to activate the Scientific Calculator when you have to calculate difficult formulas. For further information regarding the use of the scientific calculator please refer to the on-line *Reference Guide*.

- **Beeping:** It enables you to activate or deactivate the beep when a selection is made in the graphic area.
- **Echoes when creationing entities** it enables you to activate or deactivate the echo that shows the preview of the primitive that is being created. If you are using a slow PC it is better to deactivate this option.
- **Optimized Dimensioning** it enables you to automatically center the text of the dimension when creating it.
- **Bitmap view:** if deactivated, BlueCAD avoids the viewing of images in bitmap format eventually present in the drawings, therefore speeding up the redrawing process. For the moment this is of little interest and will be dealt with in *Chapter 10*.
- **Thickness view:** if this option is deactivated, each primitive will be viewed with the minimum thickness. This speeds up the visualization.
- **Fast Thickness View**: it can only be activated if **Thickness View** is also activated. In this case the primitives are viewed using only two types of thickness, minimum and the one immediately superior to it. With the latter all the primitives that have a thickness greater than the minimum will be viewed. This is a useful option for speeding up the visualization of graphic primitives.

  Note: These last three options only concern the visualization of the drawing and do not modify the drawing itself. For example, if you choose not to view the thickness, the primitives will maintain their prior values; the only effected change will concern their visualization on the screen.

- **Modify Background Color:** it enables you to modify the backgound color in the BlueCAD drawing area. You can activate by duble click.

Automatic snap and temporary snap

In the last chapter we saw how the Snap controls the movement of the cursor and how it simplifies geometrical constructions, by using the existent snap points as references. You can bind the cursor to different types of snaps, specifying points belonging to primitives or to a Grid.

We have already seen how, whenever it's necessary to repeatedly use a certain kind of snap, it's useful to make this snap automatic by using the **Snap** command. Because we will have to hook onto Grid points quite often we can set the automatic snap to the Grid

1. Select the **Snap** command from the **Horizontal Toolbar.** The **Snap Window** will appear, this indicates the different types of snap points that can be activated.
2. Select the **Grid Point** command, which binds the cursor to the points of the Grid. The contextual window will close automatically. At this point the check mark will appear on the **Snap** button. This indicates that the automatic snap is now active.

Whenever you want to deactivate the automatic snap, all you must do is reselect the **Snap** button**:** the check mark will disappear, indicating that the automatic snap has been deactivated.

Whereas if you want to activate the automatic snap on another type of point, all you have to do is click the **Snap** button twice, so deactivating it and reactivating it, then select the type of point you want from the **Snap Window**.

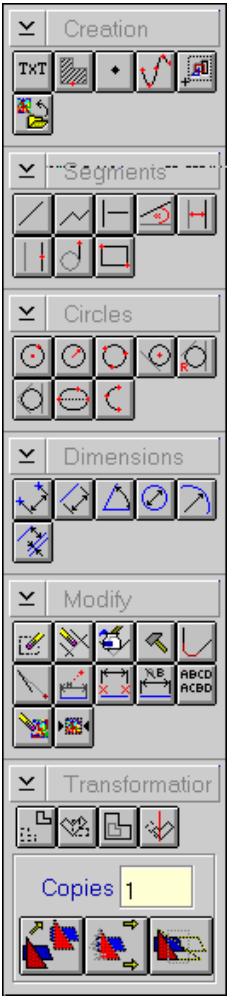
Important: Whether or not the automatic snap is active, it is always possible to use the temporary snap. Infact, when BlueCAD requires a point as input, by clicking mouse button 2, the **Snap Window** opens, from which you can make active whatever type of snap you want. This snap is temporary, because it substitutes the automatic snap, eventually present, only for the current input and then it is deactivated immediately after.

The work area

BlueCAD's graphic interface was conceived of and realized in such a way to make learning how to use it as fast as possible and to make it as efficient as possible. The contextual windows are an incredible means of speeding up the use of BlueCAD, in the same way that the grouping together of the commands on the vertical and horizontal toolbars allows a greater rationalization of the interface.

Another important aspect is that the graphic area is very vast, in such a way as to keep the drawing space as free as possible from obstacles.

Some simple considerations can be useful for rationally setting one’s own work area, therefore making the most of BlueCAD's interface:



- Get used to accessing the commands, either by the toolbars or by the **Upper Status Bar** which are preferable to the systematic use of the **Menu Bar**. To select commands from the menu, use the short keys, if available. After the initial period of learning you will realize just how efficient it is.
- It is better to draw using BlueCAD on full screen, to have the maximum drawing area possible.
- Use a background color that doesn't strain and for drawing use colors that are easily distinguishable from the chosen background color.
- The tools windows, which are opened by selecting buttons from the **Vertical Toolbar,** can be opened or closed at any time and they can remain open simultaneously. According to personal speed and access frequency, you can choose how to display the tools windows in the work area and which windows to keep open as you draw. One possibility could be to display them in column so that they form the commands tablet in the figure.

  Note: The lay-out of the windows is memorized by BlueCAD when it is shut down and re-established when the program is restarted.

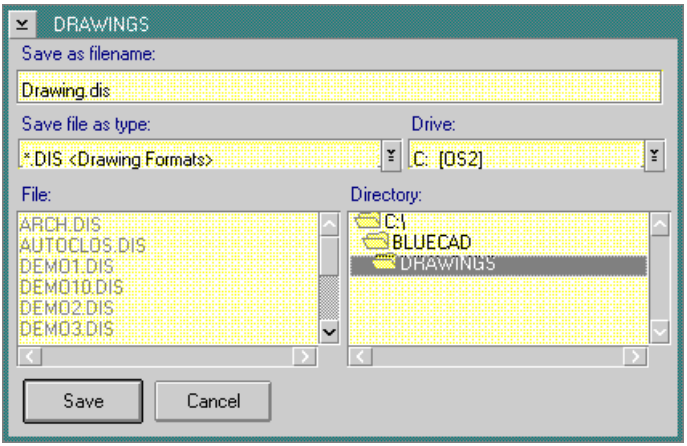
  Note: After all that has been said, the reader will presumably adopt his/her own lay-out of the tools windows or he/ she will decide on their own whether or not to keep them open. We will assume that a given tools window is closed before being used, and also that it is closed again when it has been used, even if it's not the most efficient way of operating.

The unit of measurement in the drawing

When you draw with BlueCAD, you use an adimensional unit of measurement, which specifies the distances without having to worry about setting the unit of measurement. Only when you print the drawing you specify the relation between the adimensional unit of measurement and the real reference unit, also being able to choose the desired representation scale. Let's not bother with the unit of measurement, until we look at printing.

Saving and opening a drawing

Before we start drawing, let's call the drawing PLAN.

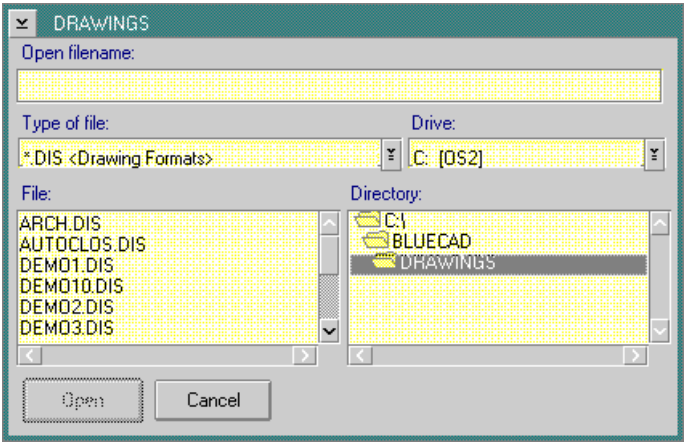


1. Select **Save as** from the menu **File.** This opens the **Save as window**
2. If you want, you can modify the assigned drive values , directory or extension in the relative entry fields.
3. Enter `Plan` into the entry field.
4. Select **Save,** as confirmation of the entered text. This saves the drawing, in the drive and in the specified directory, with the name PLAN.

Always save before leaving BlueCAD: by doing this you can pick up where you left off.

To open the previously saved drawing:

1. Select from the **File** menu the **Open** command. This opens the **Read Drawing Window**



2. Select the previously assigned drive, directory and extension values.
3. Enter `Plan` into the entry field or select PLAN from the file list.
4. Select **Open** to confirm the entered text. The drawing will be read and visualized by BlueCAD.

Carrying out the drawing

Now we will draw the walls of the room.

In drawing the walls, let's begin with drawing the external walls, using the **Coordinates Window** to specify the end points. After this, by activating the automatic snap on the Grid and using the drawn lines as relative references, you can draw the external walls. Parallelism and right-angles will be put to use by using BlueCAD's basic drawing techniques.

Here is a quick reminder about what to do if you make a mistake or have problems.

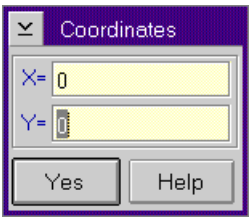
- If commands are unwanted or carried out in the wrong manner, it is possible to delete the effects of the operation with the **Undo** command. To recuperate an operation that has been undone in this way, use the **Repeat** command. The **Undo** and **Repeat** commands can be accessed from the **Edit** menu and also from the **Horizontal Toolbar.**
- If you want to remove an active command, therefore deleting the input status, select any principal command or the **Cancel Command** from the **Upper Status Bar**. This could be useful, for example, when you are executing a command and you enter the wrong input value.

Reference Lines

Now let's draw the figure walls which will later be used as reference lines. Segments 1 and 2 are the internal and external borders of the left wall, while segments 3 and 4 are the internal and external border of the lower wall. These outside walls have a thickness of 5, that is equal to the Grid step, which can be used to speed up the drawing. The dimensions of the room are 140x150. The **Coordinates Window** is used to precisely specify the extremes at the beginning.

1. Select the **Segments** button from the **Vertical Toolbar,** the **Segments Window** will appear.

2. Select the **Segments Piecewise Linear** command from the **Segments Window**. This enables you to draw the subsequent segments, which have an end point in common. The command echo signifies that the requested input is point type, which corresponds to the first vertex of the left wall.
3. Select the button **Coordinates** from the **Horizontal Toolbar**. The **Coordinates Window** will open: in field **X=** enter -80, in field **Y=** enter 70. Select **Yes** or press the **Return** key to confirm the coordinates that have been entered.



Note: To identify this point you can also move the cursor in the graphic area until the coordinates (-80,70) are indicated in the Lower Status Bar, then click. However the adopted method is faster when you have to specify, as in this case, the coordinates without having the possibility to use relative reference points.

4. The **Coordinates Window** will remain displayed, since the required input is still point type: enter the coordinates (-80,-80), the second end point of the wall, and press **Return**.
5. Enter the coordinates (60, -80), which correspond to the second vertex of the lower wall and press the **Return** key.

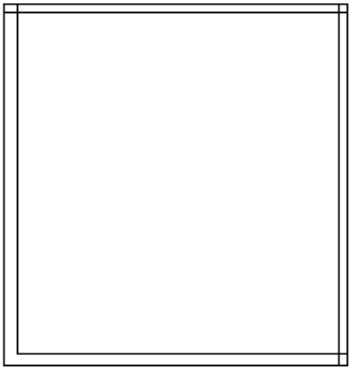
Note: Remember that the **Coordinates Window** always displays the last point that has been selected, which can be used as reference for the current selection.

To draw the internal borders of the walls let's make use of the fact that we have set the grid step to the same thickness as the outside walls and that we have activated the automatic snap on the Grid.

1. Select again the **Segments Piecewise Linear** command from the **Segments Window**.
2. To draw the internal border of the left wall, draw a segment to one grid point on the right side of the relative external border:such a segment will have the end points (-75,80) and (-75,-75).
3. To draw the internal border of the lower wall, click on point (55,-75) (i.e. one grid step above in relation to the external border).

Relative positioning

To draw the left and right walls, it isn't convenient to use the absolute coordinates or the Grid. It’s faster to use the previously created segments as references, together with the geometrical construction options available in BlueCAD. In particular we will use the option to create segments parallel to a given segment and at a certain distance from it. In this way we will obtain the figure drawing.



The upper and right walls have a thickness of 3 and must be traced in such a way that the dimensions of the room are 140x150. Let's procede creating the external borders of the walls, starting from the right wall:

1. Select the **Segments Parallel, Distance** command from the **Segments Window**. This command creates a segment parallel to another one at a certain distance from it.
2. Enter 140 in the entry field of the **Normal Calculator Window** to specify at which distance the right wall must be created. Select **Yes** or press **Return** to confirm the entered value.
3. The echo command indicate that a reference segment (in this case the external border of the left wall) has to be selected. Click on it, to create the external border of the right wall.

   Important: When selecting be careful that the center of the selection trap is on the same side as where you want to create the segment. In this case the segment must be created on the right side of the selected primitive: therefore, when you click, make sure that the center of the selection trap is on the right of the selected primitive. If a negative distance had been entered, there would be the opposite reasoning: to create, for example, a primitive on the right of the one that has been selected, such a selection must be carried out with the center of the selection trap on the left.

4. Select **Segment Parallel, Distance** again to draw the external border of the upper wall.
5. Input 150 in the **Normal Calculator Window**, then select **Yes** or press the **Return**.
6. Click on the external border of the lower wall: this creates the external border of the upper walls.

Note: At point 4, we have reselected the **Segments Parallel, Distance** command, even if already active from a previous execution: this is because the value of the distance used in the last execution, (140), will stay the same for all subsequent executions . To set a different distance, (150), it's therefore necessary to select the command again. This is a general procedure, common to all BlueCAD commands.

Using the same procedure, let's draw the internal borders of the upper and right walls, having a thickness of 3.

1. Select the **Segments Parallel, Distance** command, to now draw the external border of the upper wall.
2. Enter 3 in the **Normal Calculator Window**, to specify the distance at which the internal border must be created in relation to the external one. Click on **Yes** or press the **Return** key to confirm the value that has been entered.
3. Click on the external border of the right wall: this creates the internal border at a distance of 3 from the external one.
4. Click on the external border of the upper wall: this creates the internal border at a distance of 3 from the external one.

Modifying and finishing

First, we would like to extend the borders of the external walls to indicate their continuation beyond the dimensions of the room. For this purpose we will use the **Modify Segment** command, which allows you to quickly modify a segment that has already been created.

1. Select the **Modify** button from the **Vertical Toolbar**, to open the **Modify Window**.

2. Select the **Modify Segment** command. The command requires the selection of the segment that is to be modified, as first operand.
3. Select the external border of the left wall. Now, the command requires the new vertex of the segment as second input. The command echo will show a preview of the segment.
   Important: You must click on a point of the segment that is situated in the half of the vertex that is to be modified; in this case the upper half.
4. Click two grid steps upper in relation to the previous vertex of the segment.
5. Execute the command again, repeating points 3 and 4 and selecting the internal border.

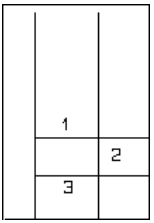
6. Do the same for the lower wall, carrying out points 3, 4 and 5 and selecting the new end points of the borders two grid steps to the right in respect to the old ones.

We have now completed the drawing of the walls of the room. Now we must to do a finishing operation to eliminate any unwanted extensions of the representation lines of the walls.

To do this the **Delete part** command is very useful since it makes many of editing operations immediate. This command, by selecting a primitive that intersects other primitives one or more times, eliminates the portion containing the selected point and having as end points the vertex of the selected primitive or its intersection point. The **Trim** command will also be used; this allows the two primitives selected to be consecutive, either by shortening or lengthening.

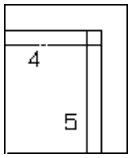
Note: Other than segments, these commands can also be applied to, using the same operative mode circles, ellipses and splines.

1. Select the **Delete part** command from the **Modify Window**, which was opened before.



2 To refinish the left wall delete the portion 1,2 and 3 by clicking, in this order, on a point belonging to portion itself.
3 Repeat the the previous operation to refinish the lower wall.

4 Select the **Trim** command from the **Modify Window**



5 To trim the internal borders of the upper right walls, click on the borders (4 and 5).

Note: The reader will have noticed how, in this case, you could have used just the **Trim** or the **Delete part** command, to carry out the above-mentioned modifications. You can try to do this as practice, after having returned, using the **Undo** command, to the situation prior to point 1.

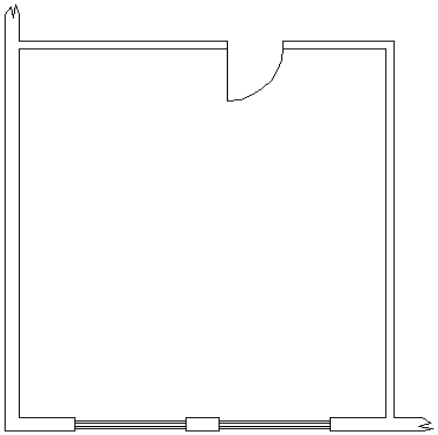
Chapter 5. Use of views and transformations

In this chapter particular attention will be paid to the view commands and to the use of transformations.

View commands are essential when you have to work on large-sized drawings. They enable you to make enlargements, reductions, to have panoramics of all sides and to visualize details separately.

As we have seen in the previous chapter, being able to quickly and easily use the existing information is fundamental for drawing efficiently with BlueCAD. We will see how the transformation commands are essential tools: managing the geometrical transformations and enabling an efficient use of drawing elements as a reference when modifying or creating graphic entities.

All this will be illustrated by drawing the cutting lines of the walls, the



windows and the door of the room and therefore finishing the drawing of the walls, which was started in the previous chapter.

Modifying the view of a drawing

With BlueCAD it is possible to carry out various viewing operations such as reducing, enlarging and modifying views. It is also possible to divide the drawing area into more windows.

We must remember that all view commands are transparent commands. This means that they can be executed at any given time without modifying the input state of the program: any previous command will remain active even after the execution of a view command. All view commands are accessible from the **Horizontal Toolbar** or, the **View** menu. Finally, it is important to remember that view commands don't modify the drawing, but only how it appears on screen.

Making an enlargement or a reduction

To enlarge a part of a drawing, you can use the **Enlargement** command. This command is usually the most used of all the view commands. It gives you the possibility to occupy the whole graphic area with the chosen part of the drawing. You can visualize the drawing with any degree of detail: this greatly facilitates working on details.

For example, if you want to enlarge the part of the drawing which corresponds to the upper right corner of the room and then to return to the starting view.

1. Select **Enlargement** from the **Horizontal Toobar** or from the **View** menu. The cursor will change shape, indicating that the area to be enlarged must be selected.
2. The rectangular area which is to be enlarged will be indicated by means of its lower left vertex and its upper right one. Click a point on the graphic area, which is slightly underneath and to the left of the upper right-hand corner of the room.

3. The command echo will show a preview of the area that is to be enlarged. Move the cursor so that the area will totally close the corner of the room.

   Note: Remember that the current coordinates, on the Horizontal Toolbar will no longer show the absolute coordinates but the ones relative to the first selected vertex. This happens to facilitates the selection of the second vertex.

4. Click on the identified point: the selected area will be enlarged and will occupy the entire graphic area.

The **Undo** command doesn't work with the view command. Use the **Zoom previous** command to undo a view command. As opposed to the **Undo** command, the **Zoom previous** command is a one-step command, that is, it only allows you to restore the very last view state.

1. To go back to the previous view, select **Zoom previous** from the **Horizontal Toobar.**

Instead, if you want to enlarge or reduce the view of the whole drawing, you can carry out a zoom operation. This varies the scaling factors with which a drawing is viewed: a positive zoom brings about an enlargement of the drawing, with factor two, a negative zoom brings about a reduction, also with factor two.

1. Select the **Zoom negative** command from the **Horizontal Toolbar**. The drawing will be reduced with factor 2.

2. By selecting the **Zoom negative** command again, a further reduction of the same scaling factor will take place.

3. You will return to the starting view by executing twice the **Zoom positive** command from the **Horizontal Toolbar** or from the **View** menu.

Panoramic and optimum views

Another possibility is carrying out a panoramic of the drawing, by creating a slide to be able to move the view area.

1. The upper walls will be entirely enlarged by selecting the **Enlargement** command from the **Horizontal Toolbar** and by using the procedure shown in the previous paragraph.

2. Select **Panoramic** from the **Horizontal Toolbar**. The command needs two points as parameters. After executing the command, the first point will move onto the second, with the consequent sliding of the viewed area of the drawing.

3. Select a point situated near the right-hand corner of the room and then a point situated near the left-hand corner. The view of the drawing will slide, in relation to the end point selected, from right to left: the right-hand corner is now viewed on the left-hand part of the graphic area, while the left-hand corner is out of view.

The view commands usually causes, as we have seen, reductions or enlargements of parts of a drawing. Therefore, it is useful to have the possibility of restoring the optimum view of the drawing, that is to view the whole drawing on the maximum scale possible. This always lets the user visualise the drawing in all its parts.

1. To view the whole drawing again on the optimum scale, select the **Zoom all** command from the **Horizontal Toolbar**.

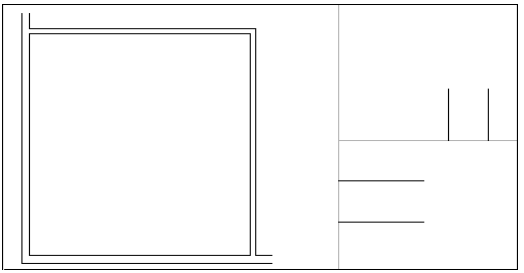
Multiple windows view

BlueCAD makes possible to manage multiple windows view. This means that the user has the possibility of having several windows available on screen at the same time; in each one of these, particular parts of the drawing can be viewed so that the desired operations can be carried out. This function allows the graphic area to be organized more efficiently and allows you to view at the same time the different drawing's parts of your interest with whatever degree of detail you want.

The possible options are: single window view , three windows view and four windows view. When passing from single window to a multi-window procedure, the view commands maintain all their previously described functional characteristics. The only variant is that they require the selection of the window they are to work on.

Note: Exceptions to this are the **Redraw** and **Zoom previous** commands. Even in multi-window procedures they don't need the selction of the window they are to work on: **Redraw** carries out the redrawing operation in all the windows on screen, while **Zoom previous** undo the last view operation, independently from the window where it has been executed.

Now let's see how to use the multi-window procedure to have the following three vindows view. This view will facilitates the drawing of the cutting lines of the walls.



1. Select the **View 3** command from the **Horizontal Toolbar** or from the **View** menu.

2. You will have the sub-division of the graphic area in three windows: there will be a larger one on the left and two smaller ones on the right, each of these will view the same previously active view.
3. Click the **Enlargement** command. Select the area to be enlarged, i.e. the part of the left wall which extends beyond the dimensions of the room (the points to be selected are approximately (-82, 70), and (73, 85)).

   Nota: The selection can be carried out by using anyone of the three windows on screen; however, it is convenient to make the selection itself easier, to use the largest window.

4. The cursor will change shape and look like a hand: this means that, compared to the single window procedure, the command now requires the selction of the window where you want to view the enlargement. Select, out of the three, the upper right window, clicking somewhere on the inside of it: the enlargement of the selected area will appear in this window.
5. Click the **Enlargement** command. Now select, as the area to be enlarged, the part of the inferior wall which extends beyond the dimensions of the room (the points which are to be selected are approximately (65,-82), and (78,-72))
6. Select the lower left-hand window, clicking somewhere on the inside of it: in this window an enlargement of the selected area will appear.

Transformations of graphic primitives

Let's go on with drawing the cutting lines for the external walls, as well as the windows and the door of the room. In doing this we will use many commands that have already been described in this chapter and we will also become familiar with BlueCAD's geometrical transformations.

BlueCAD makes available all the geometrical transformations (translations, rotations, scaling and mirroring); all of them are execute using an uniform two step operating procedure:

1. The geometrical parameters which characterize the transformation are set through the execution of one of the commands for setting transformations (**Translation**, **Rotation**, **Scale** and **Mirroring** commands)..
2. The set transformation will be carried out, through the execution of a command to make the transformation active (**Copy**, **Move** or **Stretch** commands). Therefore, a copy, movement or stretching operation will be carried out on the selected primitives, according to the transformation previously defined.

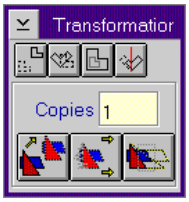
Copying primitives

Now let's draw the cutting line of the left wall, setting to 1 the step of the grid and using the **Segment Piecewise Linear** command:

1. Open the **Settings Window**, selecting the **Settings** command from the **File** menu.
2. Select the **Grid** bookmark, then enter 1 in the **Step** entry field. Close the **Settings Window** by clicking twice on its control menu.
3. Select the **Segments** button to open the **Segments Window.**
4. Click on the **Segments Piecewise Linear** icon.
5. By using the upper window view on the right hand-side, click in succession on the points: the vertex of the external border (point (-80,80)), three grid steps on the right and three up relation to the latter ((-78,83)), one more on the right and four down ((-77,79)), one on the right and five up ((-76,84)) and finally the vertex of the internal border ((-75,79)).

To draw the cutting line relative to the lower external wall, it is quicker to carry out a transformation operation rather than proceed in the same way as before: this cutting line can be obtained from the one that has just been drawn by means of either translation or rotation.

1. Select the **Transformation** button from the **Vertical Toolbar**, this opens the **Transformation Window**.



Note: The **Transformation Window** is divided in two parts: the upper part with **Translation, Rotation, Scale** and **Mirroring commands** and lower part where there are **Copy, Move** and **Stretch** commands. This subdivision reflects the previously described two step procedure for the execution of the transformations.

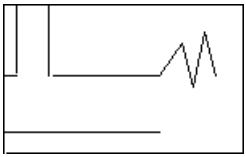
2. Select the **Translation** command. This command requires the input of two points: the translation will be equal to the distance between the two points and according to the direction identified by them.
3. Click, using the upper window view on the right, on the vertex of the outer border ((-80,80)).
4. Move the cursor into the lower window on the right. As you notice from the command's echo, the selection of the second point on a different window than the first is allowed. Click the vertex of the upper border (point(70,-75). This sets the desired translation.

5. Let's now apply the translation to the cutting line, making a copy of it: select the **Copy** command from the **Transformation Window**. The command requires the selection of the primitives that are to be copied.

6. Click mouse button 2 to open the contextual **Selection Window**.

7. Select **Selection Zone.** The **Selection Window** will automatically close.
8. Using the upper window on the right (or the biggest one), click a point which is slightly lower on the left and a second situated slightly higher on the right compared to the cutting line, so selecting all the segments belonging to the cutting line. The cutting line will be copied according to the translation set.

Moving primitives



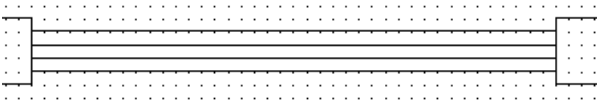
As long as the cutting line fits together with the inferior walls, it is necessary that it rotates 90 degrees in a clock-wise direction. The **Rotation** command will set the rotation transformation and then the **Move** command will carry out the rotation. The main difference between the **Copy** and **Move** commands is that the **Copy** command is a creation command while **Move** is an editing command: both carry out set transformations, but while **Copy** creates, from the primitives selected, new primitives, the command **Move** command modifies the primitives selected.

1. Select the **Rotation** command from the **Transformation window**. The command requests as input: the rotation centre (i.e. around which point you want the rotation to take place) and the rotation angle.
2. Select the rotation centre: click, using the bottom right window view, on the vertex of the upper border (point(70,-75)).
3. Enter -90 into the entry field of the **Angles Window** and press **Return** to confirm the rotation angle.

4. Now let's move the cutting line according to the set rotation: select the command **Move** from the **Transformations Window**
5. Click mouse button 2, this opens the contextual window **Selection Window** and select the **Select zone** command.
6. Using the bottom right window view (or the biggest one), click first on a point which is slightly lower and on the left, and then on a second, which is slightly higher and on the right in relation to the cutting line. The cutting line rotates in relation to the set transformation, fitting together with the lower border.
7. Now let's go back to a single window view visualizing the whole drawing: select the **View 1** command from the **Horizontal Toolbar** or from the **View** menu. The cursor will take on the form of a hand, which indicates that you must choose a window: click on a point in the biggest one.

Multiple Copy of Primitives

Now let's draw the window in the lower wall, as in the drawing. It's 20 from the left angle and 40 wide.



1 Click the **Segment Horizontal-Vertical** command from the **Segment Window**.

2 Activate the **Coordinate Window**, if it isn't already active, by clicking the **Coordinate** command on the **Horizontal Status Bar**.

3 Input the coordinates of the first extreme of the window frame, which must be 20 from the left angle of the lower wall (coordinates (-55,75)).

4 Click mouse button 2 and select the **Near Point** command from the **Snap Points Window**. This command sets the temporary snap on the point, belonging to a primitive, which is the closest to the click point.

5. Click on the external border of the lower wall: a point on its vertical will be selected. We can now use this first drawn window frame as a reference for the second.
6. Select **Segments Parallel, Distance** from the **Segments Window**.
7. Enter 40 into the entry field of the **Normal Calculator Window.** Select the **Yes** button or press **Return**.

We can now draw the horizontal line of the window by executing the **Number of Copies** command followed by the **Copy** command. The **Number of Copies** command allows you to set the number of iterations of the **Copy** command, therefore allowing the repeated application of the set transformation (translation, rotation or scaling).

Note: The **Number of Copies** command is effective only on the **Copy** command and not on **Stretch** and **Move**.

1. Select the **Enlargement** command from the **Horizontal Toolbar**.
2. Click on a point that is slightly lower on the left and a point that is slightly higher on the right in relation to the lower wall, so that it will be enlarged.
3. Set a vertical translation with the value of 1. Select the **Translation** command from the **Transformation Window**
4. Click anywhere on the Grid.
5. Click on one Grid step above. This sets the desired translation.
6. Enter the value 4 into the **N. Copies** field of the **Transformation Window**.

Note: Setting **N. Copies** from the **Transformation Window** is the same as executing the **Number of Copies** command from the **Transformation menu**

Note: It's necessary to specify the number of copies after having set the desired transformation because the setting of a transformation automatically restores the number of copies to 1.

7. Select the **Copy** command from the **Transformation Window**

8. Click on the external border of the walls. The set multiple copy will take place: the translation will be repeated 4 times.
9. Select the **Delete Part** command from the **Modify Window**.
10 To delete the extensions outside the window frames of the created horizontal segments, click on those extensions.

Using Symmetry

One of the basic techniques for using existing information is making use of any eventual symmetry that the drawing may have, by using the mirror transformation.

We now want to draw, on the lower wall, a window of the same dimensions as the one we already have and that is symmetrical in relation to the center of the wall.

1 Select the **Mirroring** command from the **Transformation Window.** The command requires the selection of two points which are part of the desired mirror: in our case it's the vertical axis which passes through the center of the wall.

2 Click mouse button 2 and select **Mid Point** from the **Snap points window**.

3 Click on the internal border of the wall: its mid point will be selected as the first point of the symmetry axis.

4 In the **Coordinates window** modify Y coordinate value by entering any value, without modifying the X coordinate. Press **Yes**. The symmetry axis has been defined.

Note: Because the decision whether to leave the **Coordinates window** active or not is left to the reader, from now on when the reader is called on to use it , it is understood that he/she activates, if it isn't already active, the **Coordinates window,** by clicking the button **Coordinate Management** on the **Horizontal Toolbar**.

5 Select the **Copy** command from the **Transformation Window**.
6 Select an area that includes the window and the frames. Click on a point that is slightly lower on the left and on a point that is slightly higher and on the right in relation to the window. This copies the window applying the set mirror transformation.

7. Select the **Delete Part** command from the **Modify window**.
8 Delete the parts of the borders that are inside the window by clicking on those parts.

Stretching Primitives

Let's draw the door of the upper wall. You will now make use of some new and very useful snap points commands. After having drawn the door, you will modify it by carrying out a stretching operation.

Note: If you want, you can now deactivate the Grid, since it isn't used at this stage, by selecting the **Settings** command from the **File** menu and by clicking on the **Visible Grid**. This speeds up the viewing operation. You can also deactivate the automatic snap by clicking on the **Snap** command in the **Upper Status Bar**.

1 Select the **Panoramic** command from the **Horizontal Toolbar**.
2 Click on point in the upper part of the graphic area.
3 Click on a point in the lower part of the graphic area, which is, more or less, on the vertical in relation to the first point. There will be a sliding of the visualized area.
4 If it's necessary, steps 1,2,3 can be repeated until the upper wall is viewed in the center of the graphic area.

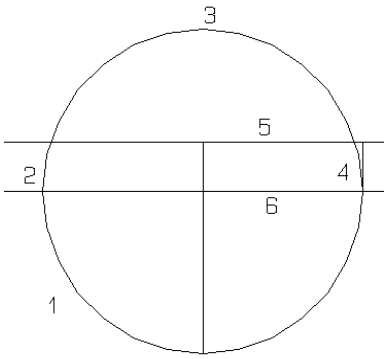
5 Select the **Circle** command, so opening the **Circle Window**.

6 Select the **Circle center, radius** command.
7 Enter the coordinates (-3, 67) of the center of the circle in the **Coordinates window** Press **Yes**.
8. Enter in 10 into the entry field of the **Normal Calculator window**. Then select **Yes** or press **Return**, to confirm the entered values of the radius.
9. Select **Horizontal-Vertical Segments** from the **Segments Window**.
10 Click mouse button 2 to open the contextual **Snap points window**.

11. Select the **Center point** command.
12. Click on any point belonging to the radius arc: its center will be selected as the segment's first end point.
13 Click mouse button 2, and select the **End Point** from the contextual **Snap points window**
14. Click on a point belonging to the lower semi-arc of the circle: its end point will be selected as the segment's second end point.

Note: To select this point it would also have been possible to have used the **Near Point** snap command. Note that the **End Point** command, when applied to a circle, selects, among the four intersection points of the circle with their own horizontal and vertical diameters, the one closest to the click point.

15 Click mouse button 2, then select **Center point** from the contextual window **Snap points window**
16. Click on a point belonging to the circle, its center point will be selected.
17. Click on mouse button 2, and select **Near point** from the contextual **Snap points window**
18. Click on any point belonging to the external border of the wall: the point belonging to it and situated on the vertical axis will be selected.



19. Click mouse button 2, then select **Intersection point** from the contextual window **Snap points window**
20. Click a point belonging to the circle and on a point of the internal border belonging to its right hand part: the intersection point between the circle and the internal border will be selected.
21. Click on mouse button 2, then select **Near point** from the contextual **Snap points window**
22. Click on any point belonging to the external border of the wall: the point belonging to it, situated on the vertical axis, will be selected.
23. To Open the **Modify window,** select the **Modify** button; then select the **Delete part** command to delete the part of the circle that is external to the room.
24. Click on any four points of the circle which belong respectively to the arcs 1, 2, 3 and 4 of the figure; click on any two points which belong to the segments 5 and 6

    Note: If you have any difficulty in the selection, carry out the **Enlargement** or **Zoom positive** commands to have an enlargement of the selected primitive.

Let's now suppose that you want to move the door that you have just drawn more to the left, by about 3. How can this be done without having to modify either the door or the wall? You can use the **Stretch** command, which carries out a stretching operation on the selected primitives applying the set transformation. This operation, for the primitives completely internal to the selected area, is the same as carrying out a **Move** command; instead, the primitives that have an internal end point and an external one to the selected area, will be "stretched": that is they will be modified, taking into account that the external extreme will remain unchanged, while the internal one will have the set transformation applied to it.

1. Select the **Translation** command from the **Transformation window**.
2. Click on any point in the graphic area.
3. Increase by 3 the value of the X coordinate present in the **Coordinates window** then press **Yes** or **Return**. This sets a horizontal translation towards the right by 3.
4. Select on the command **Stretch** from the **Transformation window**. The command requires the selection of the primitives that are to be stretched.
5. Select a point that is slightly lower and on the left and in a point that is higher on the right in relation to the door, so selecting the door and the door jambs. In this way a stretching takes place in relation to the previously set translation transformation. The primitives that make up the door will be translated, while the borders of the walls , that are straddling the selection box, will be stretched: the part of the wall to the right of the door will be shortened; the part to the left will be lengthened.

Modifying the dimensions of objects

Let's take a closer look at the use of transformations in BlueCAD, by considering the following problem: we want to modify the dimensions of the door, trying to modify as little as possible of the existing geometry.

One solution could be to transform the drawn door into a block, and then to apply the scale you want to it, so modifying its dimensions. This will be dealt with in *Chapter 7*, when the use of blocks will be explained. Anyway, it isn't necessary to have to resort to blocks to solve this problem: you can find a solution by making use of the scale transformations.

Let's double the dimension of the door, keeping the position of the left door jamb fixed:

1 Select the **Scaling** command from the **Transformation window**. The command requires a scaling point, which is the scaling reference point, and a scaling factor, which is the factor that the dimensions must be scaled.

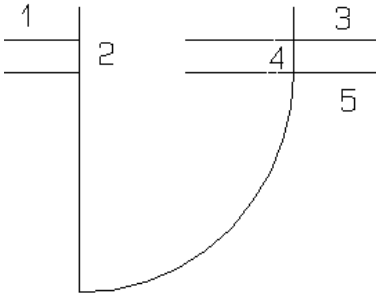
Note: The scaling factor can be greater or lesser than 1, it all depends on whether you want to enlarge or reduce. It can also be negative (you will obtain a mirroring in relation to the scaling point) but it cannot be zero.

2 Click on mouse button 2 and select **Center point** from the **Snap points window**.

3 Click the arc of the door: the center will be selected as the scaling point.

4 Enter the value 2 and press **Yes** to confirm the value of the scaling factor.

5 Select the **Move** command from the **Transformation window**
6 Select an area that includes the door and the door jambs, by clicking on a point slightly lower and on the left and a point slightly higher on the right. The door will then be modified in relation to the set scaling factor: the dimensions will be doubled, keeping the left door jamb fixed.
7 Select the **Trim** command from the **Modify window**.



8 To cancel any unwanted extensions of the door jambs, click repeatedly on the points belonging to the following segment portions: 1, 2; 3, 4; 5, 4.
Note: For this operation you could have also used the **Delete part** command.

From all that has been said in the previous paragraph, one could be led into thinking that applying the command **Stretch** twice, once to the right door jamb using a translation of 10 towards the right, and once at to the right vertex of the arc using a translation of 10 towards the bottom, you could have obtained the same result, with the added advantage of not having to retouch the door jambs. In reality this isn't true: you wouldn't have obtained an isometric transformation: the proportions wouldn't have been maintained as they are. This means that the arc of the door would have to be redrawn, because its radius would not be 20 but a greater radius.

Considerations on viewing and Selection window

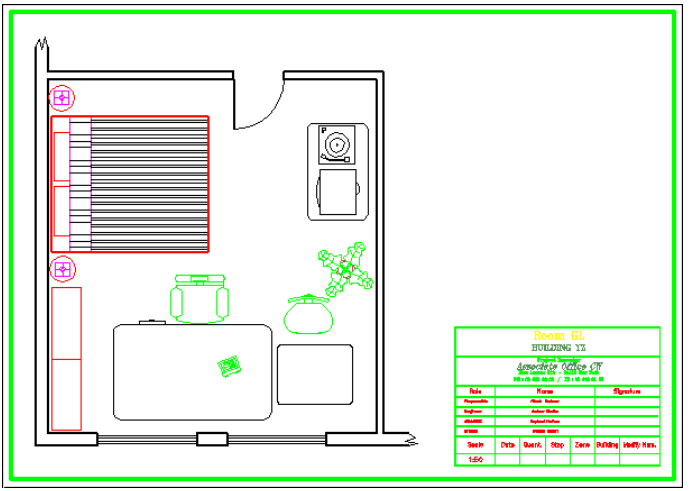
At the end of this chapter it is necessary to point out certain things about the use of the contextual **Selection window.** We have seen how this window enables the access to the selection commands: you can select the primitives within a rectangular area, the primitives that are outside a rectangular area or all the primitives. It is important to underline that these commands work on the primitives that are in the current window view, in this way giving a greater possibility of selection. For instance, by carrying out the **Select all** command, not all the primitives of the drawing are selected but only all the primitives currently visualized.

Chapter 6. General attributes, information and text use

BlueCAD primitives are characterized by attributes associated to them. General type attributes are relative to shared characteristics of every type of graphic primitive, while specific attributes are relative to particular primitive type. We'll attend to the general attributes first of all, keeping in mind however that the method for operating on the attributes in BlueCAD is common to the general attributes as to the specific ones. The latter will be illustrated in the course of the following chapters, contextually to the primitives to which they refer, beginning in this chapter, when it pauses on the use of the text.

In addition, we'll see, how to obtain information on the entities of the drawing, how to find out the current status of the attributes and how to get other general information.

In the course of the chapter the cartouche will be added to the plan.



Use of the attributes

General attributes (color, thickness, line type, and layer) exist in BlueCAD, that is common to every type of graphic primitives, and specific attributes of the dimension primitives, texts and hatchings. The commands to operate are accessible, as you remember from *Chapter 3,* indifferently from the **Attributes** menu or from the **Upper Status Bar**.

There are two ways to operate on the attributes of the graphic primitives:

- define attributes mode
- modify attributes mode

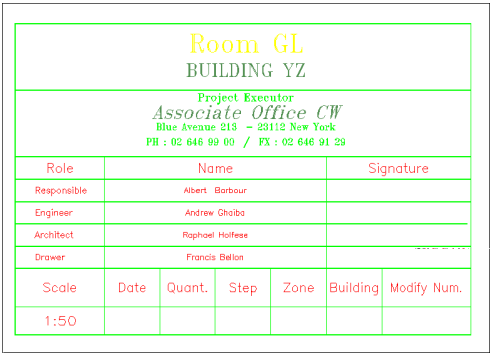
These are the two alternative modes that can be activated by means of the **Define** and **Modify** commands present in the **Attributes** menu, or through the bistable **Define-Modify** button present in the **Upper Status Bar**. The define mode enables the setting of the values of the attributes in such a way that the values become assumed by the primitives that they will create. The modify mode instead enables the modification of the values of the attributes of the primitives already created.

Note: When opening a new drawing, the attributes are initialized to their default value. If the user defines new attributes values, these values will remain those assumed for that drawing, also at the next start of BlueCAD, until different ones aren't defined.

The procedures to define and modify the attributes, detailed in the next paragraphs, are the common modify and define procedures of all the attributes, either general (color, thickness, line type, and layer) or specific attributes of texts, dimensions and hatchings.

Example of attributes definition

Let's draw a 82X56 cartouche, on which the mandatory writing of the drawing will be placed. While executing this, refer to the drawing in the figure. Until now the default color and thickness were used to draw (respectively the color white and thickness 1): let's now define the color as green and thickness as 2.

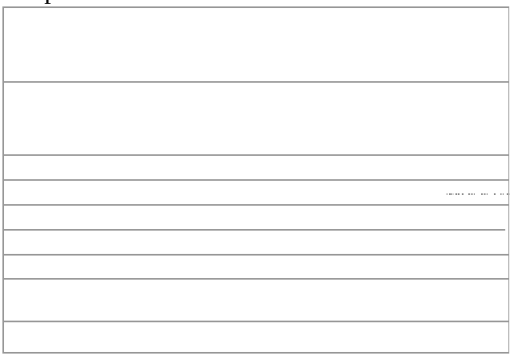


1. Activate the define attributes mode, if it isn't already active, by selecting the **Define-Modify** button from the **Upper Status Bar**. The define mode is active when the button shows the `Define` writing.
2. Select the **Color** button from the **Upper Status Bar**. This opens the **Color Window**.

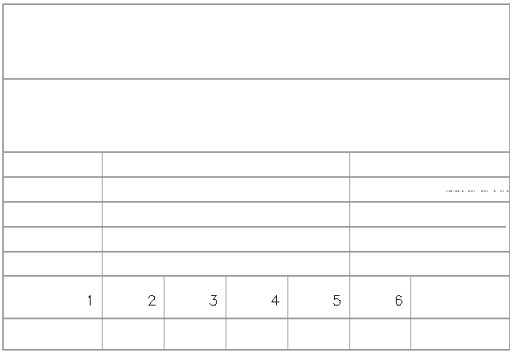
3. Select the color green, the fourth button on the left of the first line. The **Color Window** closes automatically. All the primitives created from now on will be green, until you define a different color by using the same procedure.
4. Select the **Thickness** button from the **Upper Status Bar**. This opens the **Thickness Window**.

5. The thicknesses brought into the window are in increasing order, from the left to the right. Therefore, to select thickness 2, click on the second button on the left. The **Thickness Window** closes automatically.
6. Select the **Negative Zoom** command from the **Horizontal Toolbar**.
7. Open the **Segments Window** and select the **Rectangle** command.
8. Select the lower left vertex of the cartouche (89, -88) using the **Coordinates Window**.
9. Select the upper right vertex of the cartouche (171, -32) using the **Coordinates Window**.

10. Select on the **Enlargement** command. Select the rectangle of the cartouche as the area to enlarge, by clicking on a point placed slightly more at the bottom and to the left and then on a point slightly more on top and to the right than the cartouche itself.
11. Activate the Grid, if it isn't already active, with step 1, by selecting the **Settings** command from the **File** menu and clicking on **Visible Grid**. Also activate the automatic snap on the Grid, by clicking on the **Snap** button and selecting the **Grid Point** command from the **Snap Points Window**. The automatic snap on the Grid and the echo of the coordinates in the **Lower Status Bar**, will facilitate you to individuate and select the points of interest.



12. Select the **Segment Parallel, Point** command. Select, in succession, the upper side of the cartouche and then a point, inside the cartouche, placed 13 grid lines lower. A segment parallel to the upper side of the cartouche will be created and a distance of 12 from it.
13. To draw the other seven horizontal segments of the cartouche: click, with respect to the last created segment, on 13 grid steps down, then 4 down for five times, finally 5 down.
14. To draw the vertical segments of the cartouche, define again the thickness 1: select the **Thickness** button from the **Upper Status Bar**, then select thickness 1 from the **Thickness Window** by clicking on the first button on the left.



15. To draw vertical segments of the cartouche select the **Segment Horizontal-Vertical** command; thus drawing the 6 segments having the following end points: a) segment 1: first end point 13 grid steps to the right with respect to the lower left vertex of the cartouche and second end point the point of intersection with the third, from the top, horizontal segment of the cartouche b) segments 2, 3, 4, 5, 6,: 10 grid steps to the right of the grid with respect to the last segment created and the intersection of the third, from the bottom, horizontal segment.
16. Deactivate the Grid by selecting the **Settings** command from the **File** menu and clicking on the **Visible Grid**. Also deactivate the automatic snap on the Grid by clicking on the **Snap** button.
17. Return to the original visualization by selecting the **Zoom Previous** command from the **Horizontal Toolbar**.

Note: If the **Fast thicknesses view** option has been activated from the **Settings-General Page Window**, then all the primitives appear with the minimum thickness. This option enables BlueCAD to redraw more quickly. Deactivate this option if you want to see the real thicknesses of the entities.

Example of attributes modification

Now we want to modify the thickness of the walls and of the door to the value 2.

Important: When speaking of attributes, you refer indifferently to their name (for example color green) or to the numerical value that represents them (color number 4): this numerical value corresponds to the value that you must specify when you carry out a modifcation of the attribute. For the complete correspondence between the attributes and the corresponding numerical values see the on-line *Reference Guide*.

1. Activate the modify attributes mode, if it hasn't already been activated, by selecting the **Define-Modify** button from the **Upper Status Bar**. The modify mode is active when the button shows the `Modify` writing.
2. To open the **Thickness Window**, select the **Thickness** button from the **Upper Status Bar**.
3. The thicknesses brought into the window are in increasing order, from left to right: to select the thickness 2, click on the second button from the left. The **Thickness Window** closes automatically.
4. By selecting the segments making up the walls and the door, they are modified by a thickness 2. Help yourself, wherever necessary, with enlargements, to facilitate the selection.

The procedure shown for the thicknesses is the general procedure to follow also in the case of modifying the other general attributes (color, line type, layer) and also the specific texts, dimensions and hatchings attributes.

Obtaining information

You often find yourself in need of obtaining information concerning the drawing being executed: information on the current attributes (what is the defined color?), on the attributes of the drawing primitives (what thickness does that primitive have?) or on their geometric characteristics

(what is the center of that circle?); on distances, angles and coordinates (what is the distance between these two points?) and even on the drawing in its completeness (how many primitives are there in the drawing? and how many blocks?). Unlike the manual drawing, this and other information can be obtained in an immediate way: we will now see how and its usefulness in the drawing process.

Information on the defined attributes

To know in any moment which are the defined values of the general attributes of color, line type and thickness, it is sufficient to refer to the **Current Line Attributes Window** in the **Upper Status Bar**. It visualizes a segment with the current attributes: for example, it is now drawing with the minimum thickness, with the color green and continuous line type.

Next to it, on the right, the **Layers Window** shows the current level: `Initial layer 1`. We will return to this window in the next chapter, which specifically deals with layers.

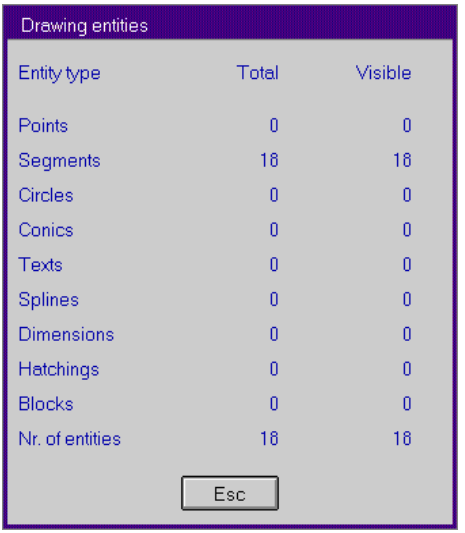
In addition, information on the current numerical value of any attribute is visualized in the **Lower Status Bar**, when you select the define or modify attribute command.

1. Select the **Thickness** button, for example, from the **Upper Status Bar**: the message `Current value 1` appears in the **Lower Status Bar**. Close the **Thickness Window** by clicking on any point in the graphic area or by pressing the **ESC** key.

Information on the drawing

To have information on the current drawing use the **Drawing Information** command, accessible in the **Information** menu:

1. Select the **Drawing Information** command from the **Information** menu.



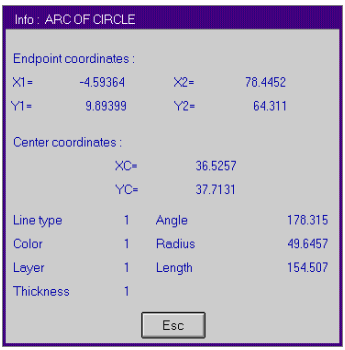
2. The window that appears lists the primitives present in the drawing, also specifying how many of these primitives are currently visible.
   Note: This last information is particularly useful to know if what is visualized and for not "forgetting", as sometimes happens, about the primitives temporarily out of visualization.

Information on the primitives

To have information on one graphic entity, use the **Graphic Entities Information** command, accessible in the **Vertical Toolbar** or from the **Information** menu: this provides information on the attributes values of the entity as on its geometrical characteristics. It behaves like a transparent command if carried out when another command is active, while it is a main command in the opposite case: this is convenient when information on more entities in succession is requested. If you wish to know, for example, the characteristics of the arc of the door and also of the external side of the upper wall:

1. Select the **Cancel** command from the **Horizontal Toolbar**. In this way the input status is reset and the active command is deactivated: the **Graphic Entities Information** command will behave like a main command and after the first execution, it will remain active for further executions.

2. Select the **Graphic Entities Information** button from the **Horizontal Toolbar**.
3. Click on the arc of the door.
4. The window that appears visualizes the geometrical characteristics of the selected primitive (in this case the type of primitive, the angle subtended, the length of the arc, the radius, the coordinates of the center and the end points) as well as its attributes (in this example the color, thickness, line type, layer of the arc). To close the window click on the **ESC** pushbutton or press the **ESC** key on the keyboard.



5. Click on the external border of the upper wall: the window that appears visualizes the geometrical characteristics as well as its attributes. To close the window click on **ESC**.

Information on dimensions, areas and perimeters

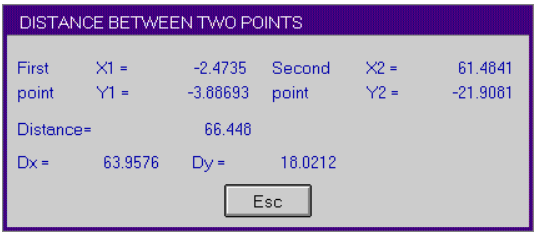
We have seen, in the previous paragraph, how it is possible to obtain geometrical information on the drawing primitives. It is often necessary, however, to obtain geometrical information in relation to points belonging to different primitives or not belonging to any primitive: for example, you may want to know what is the distance between two primitives or to measure a certain angle. You can easily guess that to create a primitive passing for the points of interest to then request information on itself, is not the most rapid way to operate. The dimensions are the general tools provided by BlueCAD for the purpose of measuring and providing the dimensions of the objects; *Chapter 9* deals with these specifically.

If instead you want to obtain information on areas and perimeters, you can use the hatching primitive, which we have seen in *Chapter 2*. By means of the **Hatching** command, it is enough to select only one side of a perimeter because the closed path, if existing, is automaticaly recognized, and the relative area is hatched. At this point by using the **Graphic Entities Information** command, seen in the previous paragraph, and by selecting the created hatching, you obtain the measurement of the area and the perimeter, and also other information like the *barycenter* of the area and the *barycentric moments of inertia*

Information on distance between points and on coordinates

Without having to use the dimensions, you can have information regarding the distance between two generic points in the graphic area, by using the **Points Distance** command; with this command you can also know the coordinates of a point of interest. **Points Distance**, like the **Graphic Entities Information** command used previously, is a transparent command if executed when another command is active, while it is a main command in the opposite case: it is convenient when you want to measure more distances in succession. Let's suppose we now want to know what the thickness of the upper wall is:

1. Select the **Points Distance** command from the **Information** menu.
2. Click mouse button 2 and select the **Mid Point** command from the **Snap Points Window**
3. Click on the external side of the upper wall, selecting in this way its mid point.
4. Click mouse button 2 and newly select the **Mid Point** command from the **Snap Points Window**
5. Click on the external border of the upper wall: a window showing the coordinates of the selected points and also their distances is visualized.



Text use

The insertion of the text in drawings of a certain complexity is one of the more laborious operations and often among the most boring. At the same time the notes are extremely important in order that the drawing may be legible and comprehensible. With BlueCAD you have the possibility to quickly insert the text, in addition to being able to control all the specific attributes to give it a professional look.

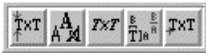
The text use will be illustrated, inserting the necessary writings of the drawing in the cartouche. This will give way to explaining the basic procedures in positioning and alligning the texts, in controlling the text boxes, and managing the attributes.

The text attributes

The texts are characterized , other than by the general attributes of color, thickness, line type, and layer, by the following specific attributes:

- Text font
- Height
- Slant
- Base to height ratio
- Origin

In BlueCAD, as we have already reminded you, there is a standardized method for operating on the attributes, either general attributes or specific ones: the procedure for defining and modifying the attributes that was shown at the beginning of this chapter, therefore applies also to the aforementioned attributes of the texts.



A command accessible through the **Text Attributes Window** corresponds to each of them, for its modification and definition.

This window can be viewed by selecting **Text Attributes** from the **Attributes** menu, or selecting the **Text Attributes** button from the **Upper Status Bar**.

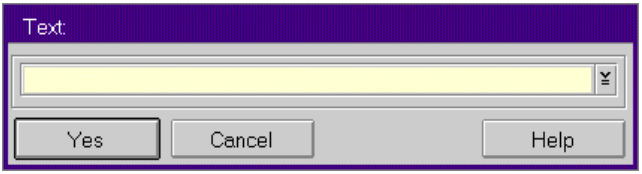
Text font

The **Text font** command enables the setting of the desired *font* by choosing among the ten available BlueCAD's fonts. Combining the attributes of slant and text base to height ratio, you can obtain other customized styles. These ten font are identified by a number: the fonts from 1 to 5 correspond to proportional fonts, while the fonts from -1 to -5 are the analogs of non proportional type.

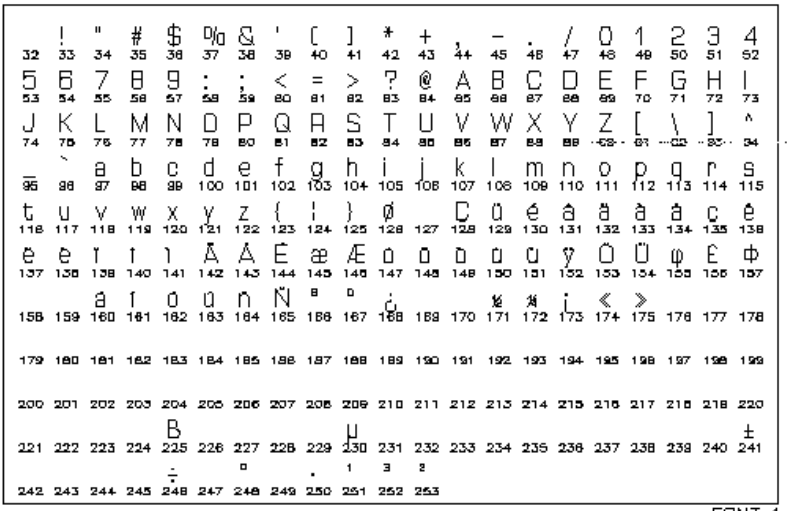
Font number 1	Font number -1
Font number 2	Font number -2
Font number 3	Font number -3
Font number 4	Font number -4
Φοντ νυμβερ 5	Φοντ νυμβερ -5

Proportional font means that the width of the font characters is not constant, but varies from character to character, permitting a greater legibility. A non proportional font is instead a font having a fixed character width. This is useful for setting the texts and numbers into columns. In using one font rather than another, remember that fonts 1 and 2 (and the non proportional correspondents -1 and -2) are simpler fonts; 3, 4 and 5 (and the non proportional correspondents) are more complex and therefore they slow down the redraw operation. The font assumed by default is font 1. The fonts used are extended type fonts: thus there are special symbols and non alphanumerical characters available. The drawings in the figure show the tables with the characters present in fonts 1 and 5 and the corresponding decimal *ASCII code* (the other fonts have the same characters as font 1). The ASCII code is an identification code of the characters: the characters can be entered in the **Strings Window** through the sequence **ALT +** *ASCII* code, where the code is entered using the little numerical keyboard placed at the right on the keyboard. For example, if you want to use the character >>

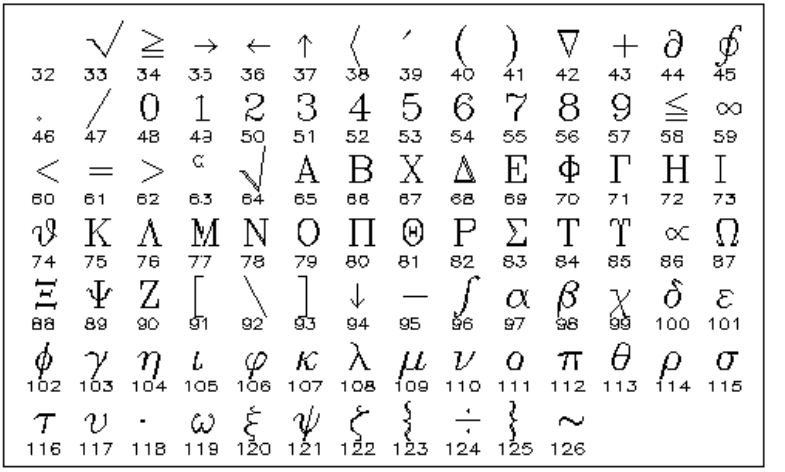
1. Click on the **Creation** button from the **Vertical Toolbar**
2. Click on the **Text** command. The input **Strings Wimdow** opens.



3. Enter **ALT + 175,** where**,** as you see in the table above, 175 is the code for the character >>. The desired symbol is now digited.



FONT 1



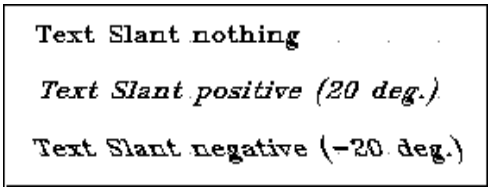
FONT 5

Text Height

The **Text Height** command enbles you to modify the height of the text, without changing the proportions of the text itself.

Text Slant

The **Slant** command enables you to modify the slant of the text. The angle can be positive (slant forward), negative (slant backward) or zero (default value).



Base to Height Ratio

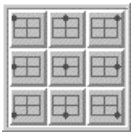
The **Base to Height Ratio** command enables you to modify the ratio between the base and the height of the text characters: if we keep the text height fixed, the text lengthens if such a relationship increases and vice versa. If the text doesn't occupy the desired text box, you can change the value of this attribute and of the text height to bring the text box (the rectangle containing the text) to the desired dimensions.

The default value of this attribute is 0.7.

Base to Height Ratio 0.5

Base to Height Ratio 1

Text origin



The **Text origin** command enables you to modify the point of the text box that is assumed as the origin of the text itself. Selection of the **Text origin** command you open the **Text Origin window** from which you can choose, among the 9 snap points of its box, the text origin point.

See the on-line *Reference Guide* for the correspondence between the snap points of the text's overall box and the numerical value of the attribute of text origin. The value of default of the text origin is 1 (lower left vertex).

If, for example, you choose the lower right vertex of the box as text origin point, this point will be used as reference point when positioning and selecting the text.

Important: The text origin point is one of the snap points available in BlueCAD: it is, in fact, possible to activate the snap on the text origin point by selecting the **Origin Point** command from the contextual **Snap Points Window**: such a command enables the selection of the origin point of hatchings, dimensions, blocks, and texts.

Defining the text attributes

When, as in this case, you must draw the texts using different styles, it is usually faster to proceed by inserting the texts according to a defined general style, without worrying too much if the text boxes aren't exactly those desired. You can proceed in refining the text later.

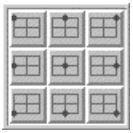
Let's define a style with the following attributes: font 2, base to height ratio 0.55, text height 1.6. Let's also define the current color as red.

1. Activate the define attributes mode, if it isn't already active, by selecting the **Define-Modify** button from the **Upper Status Bar**.
2. Select the **Text Attributes** button from the **Upper Status Bar** to open the **Text Attributes Window**.
3. Select the **Text font** command, the third button from the left. Enter 2 in the **Normal Calculator Window** field. Select **YES** or press the **Return** key to confirm the entered value.
4. Select the **Text Attributes** button again and then the **Base to Height Ratio** command. Enter 0.55 in the **Normal Calculator Window** field and select **YES**.
5. Select the **Text Attributes** button again and then the **Text Height** command. Enter 1.6 in the **Normal Calculator Window** field and select **YES**.
6. Select the **Color** button from the **Upper Status Bar** to open the **Color Window**: select the color red, the third button from the left of the first line.

Alligning the text

Let's start to insert the text in the cartouche, alligning it along the vertical passing for the center of the cartouche itself.

1. Let's draw a reference vertical segment. Open the **Segments Window** and select the **Segment** command.
2. Click mouse button 2 and select the **Mid Point** command from the **Snap Points Window**. Click on a point belonging to the upper side of the cartouche. Select the **Mid Point** command again and click on a point belonging to the lower side of the second square of the cartouche.
3. To easily select points belonging to the drawn reference vertical, let's set the automatic **Near Point** snap: select the **Snap** button and, from the **Snap Points Window**, select the **Near Point** command.



4. To position the text, centering it on the vertical we define, as text origin point, the center of its overall box. Select the **Text Attributes** button and then the **Text Origin** command. Select the second button of the second line in the **Text Origin Window**.
5. Select the **Text** command in the **Creation Window** to Open the **Strings Window**.
6. Digit the text `Room GL` in the **Strings Window** entry field then press **YES** or **Return**.
7. Specify the positioning point of the text by clicking on a point belonging to the reference vertical and having the approximately coordinate Y= -36. The automatic near point snap enables the exact selection of a point belonging to the vertical.
8. On the guideline of points 6) e 7), position the texts `Building YZ` in the first square, then `Associates office CW, Blue Avenue 213 - 23112 New York, PH : 02 646 99 00 / FX : 02 646 91 29 i`n the second square.
9. Select the **Delete** command from the **Modify Window** and delete the reference vertical, selecting it.

Centering the text

Now let's see how to position a text at a center of a square in a table:

1. Define the center of its overall box as the text origin point.
2. Indiviuate the center of the square. It can be done by:
   - Drawing the two diagonals (segments joining the opposite vertices) of the square: their intersection is the center of the square.

   - Drawing one of the two diagonals and selecting the mid point, which is the center of the square.

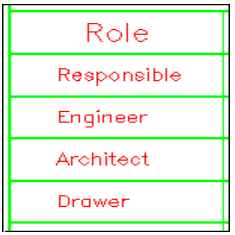
3. Select the center of the square as a positioning point of the text.
   Note: In the procedure illustrated previously, we recommend the use of snap commands from the **Snap Points Window:** in particular the snap **Intersection Point** command to select the vertices of the square, when you draw the diagonals, and the **Mid Point** command to select the mid points of the diagonals themselves.

4. Delete the reference diagonals.

The texts of the figure are inserted, applying the illustrated procedure.

Modifying the text

Another technique for alligning the texts is that of positioning a reference text to make successive copies of. This way of working, which we will now adopt to allign the texts of the figure, will permit us to describe how to modify a text.



1. Let's draw a reference vertical segment, to position the text `Responsible` in the highest square in the figure. Open the **Segment Window** and select the **Horizontal-Vertical Segment** command. Select a point on the upper side of the square, having around X=93 as coordinate, and the second end point belonging to the lower side of the same square.
2. Select the **Text Attributes** button and then the **Text Origin** command. Select the first button on the second line from the **Text Origin Window**.
3. Select on the **Text** command in the **Creation Window**. The input **Strings Window** opens: enter `Responsible`, then push **YES**.
4. Click mouse button 2 and select the **Mid Point** command from the **Snap Points Window** Click on a point belonging to the reference vertical, so positioning the text on the mid point of it.
5. Set a vertical translation, downwards, having the same height of the square. Select the **Transformation** button, then, from the **Trnasformation Window,** select the **Traslation** command.
6. Click mouse button 2; select the **End Point** command from the **Snap Points Window** and click on the upper end point of the reference vertical. Repeat the previous operation by clicking on the lower end point of the vertical.
7. Execute a multiple copy of the text `Responsible`: enter the value 3 in the **Number of Copies** field in the **Transformation Window**, select the **Copy** command and click on the text `Responsible.` Such a text is copied in the lower squares, obtaining the desired allignment.

8. Click on the **Delete** command from the **Modify Window** and delete the reference vertical by selecting it.

9. Now that the texts have been copied and correctly positioned you can now modify them. Select the **Modify Text** command from the **Modify Window** and click on the string to be modified. The **Strings Window** will appear: enter the new string in the entry field, press **YES** to substitute the old string with the new one. Modify the texts created previously according to the new texts `Engineer`, `Architect`, `Drawer`.

The positioning of the texts is now completed. Note how the `Responsible` string has a length greater than its square: in the next paragraph we will see how you can resize the texts by modifying their attributes.

Modifying the text attributes

We want to modify the attributes as some of the texts created, to make them equal to those in the figure:

1. Activate the modify attributes mode by selecting the **Define-Modify** button from the **Upper Status Bar**.

2. Select the **Text Attributes** button from the **Upper Status Bar**. This opens the **Text Attributes Window**.
3. Select the **Text font** command, the second button from the left. Enter 3 in the **Normal Calculator Window** field. Select **YES** or press the **Return** key to confirm the entered value.
4. Click mouse button 2 and select **Select Zone** from the contextual **Selection Window**. Select a zone including all, and only, the texts from the two upper squares: they are modified to font 3.
5. Select **Text Attributes** button. Select the **Text Height** command. Enter 1 in the **Normal Calculator Window** field. Select **YES** or press the **Return** key to confirm the entered value.

6. Click mouse button 2 and select the **Select Zone** command from the contextual **Selection Window**. Select a zone including all, and only, the texts relative to the `Role` and `Name` columns. The texts are now opportunely resized
7. Repeat steps 5 and 6, modifying the height of the texts to the following values: text `Room GL`, `Building YZ` to 3.4, `Associates office CW` to 2.4, `Project executor` to 1.4, address to 1.2.
8. Select **Text Attributes** again. Select the **Slant** command. Enter 15 in the **Normal Calculator Window** field. Select **YES**. Select the `Associates office CW` text to modify its slant.
9. Select the **Color** button from the **Upper Status Bar**. This opens the color window: select the color green, the fourth button from the left on the first row of the window. Select the strings `Room GL`, `Project executor` and also the address, modifying them in green.

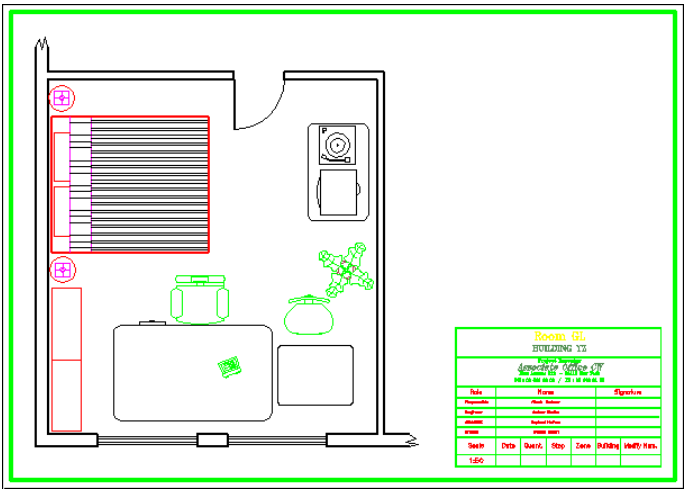
Chapter 7. Organization of the drawing: blocks and layers

BlueCAD contains two very important tools for the organization of the drawing: blocks and layers. Learning how to use these two tools in the best possible way is very important, because it contributes to speeding up the drawing process and to also greatly increasing the efficiency.

Using the layers allows you to draw in "tiers". This means that you can separate and organize the information on different layers; it's as if the drawing is presented on different sheets of tracing paper one on top of the other. Some of the advantages of this are the possibility of separating information so that it can be found easily, being able to view the drawing and work on it by parts.

Through use of the blocks the most of the work is done once, to then be reused: for example, the door and the window drawn in the last chapter, can, after having been memorized as blocks, reused and shared by other daftsmen. To efficiently use BlueCAD, the drawing's parts that are used frequently should be stored in a blocks library: a mechanical user should have a library containing screws, bolts, cog wheels and other things that are often used, an electronic engineer should have libraries of electronic components; an architect should have block libraries of doors, windows, furniture. In this way most of the drawing process can be carry out just positioning blocks.

Throughout this chapter the plan will be completed adding furnishing to the figure.



Use of the on-line *Reference Guide* is advisable for more details.

Using layers

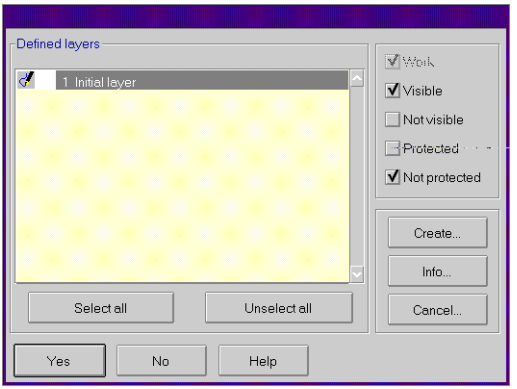
Each graphic primitive is characterized by general attributes of color, line type, thickness and layers. This means that each primitive belongs to its own layer, just like a color, a thickness and a line type. You can operate on the layer attribute in the same way as on all the other attributes, according to the modifying or defining attribute mode that was dealt with in the last chapter. These are the alternative modes that we have learnt to know and that can be activated by using the bistable **Define-Modify** button present in the **Upper Status Bar**.

The difference, in respect to the general attributes, is that, excluding the initial layer, there aren't any other predefined layers; it is the user who creates the layers he/she wants. Once the desired layers have been created, you can move the graphic primitives from a layer to another or decide, when drawing, on which layer to position the new primitives.

Creating layers

Each layer is identified by a name and an associated numeric value. Until new layers are created there will only be one predefined layer, the layer number 1 called the INITIAL LAYER: therefore this is the default layer, and the graphic primitives that we have drawn up to now have been created on it. We now want to organize on two different layers the border of the drawing and the cartouche on one side and the plan of the room on the other. We also want to create a third layer where we can position the furniture. In this way we can have the possibility to view and to operate separately on parts of the drawing that are separate from a logical point of view.

1. Select the **Manage Layers** button. This opens the **Manage Layers Window**.



2. This window lets you create and delete layers, and allows you to manage the layer's characteristics. Click the button **Create** to create a new layer.

3. This opens the **Layers Creation Window**: enter `Cartouche Layer` into the **Description** field and the value 2 in the **Number** field, as the identifier of the layer. The two options present, **Visible** and **Protected,** let you define the characteristics of the layer. The **Visible** option controls if the primitives of the layer have to be visualized or not. The **Protected** option controls if the primitives of the layer are protected or not from editing. These options are made active and are deactivated by clicking onto the corresponding buttons: an option is active if there is a check mark on the relative button. Select **Yes** to confirm the creation of the layer.
4. Click the **Create** button, then enter `Furniture for the room` into the **Description** field and the value 3 into the **Number** field, as its own identifier. Select **Yes** or press **Return** to confirm the values that have been entered.
5. Select **Yes** or press **Return:** at this stage there are now two new layers available.

Modifying the layer of the primitives

Let's move the cartouche onto the Layer of the cartouche. We can modify the layer attribute of the primitive of the cartouche from 1 (Initial layer) to 2 (Cartouche Layer), in the following way:

1. If the define attribute mode isn't already active then activate it, by selecting the **Define-Modification** button from the **Upper Status Bar**. The modify mode is active when you see the writing `Modify` on the button.
2. Select the **Layer** button from the **Upper Status Bar**. This opens the **Calculator Window**.
3. Enter 2 into the entry field, which identifies the layer of the cartouche. Then press **Yes** or the **Return** key to confirm the value.
4. Click with mouse button 2 and select the **Select Zone** command from the contextual **Selection Window**. Then select a region that includes the cartouche, by clicking on a point that is slightly lower than its lower left vertex and on a point that is slightly higher than its upper right vertex.
5. In this way the layer of the cartouche primitives has been modified from Initial Layer to Cartouche Layer. To verify this, click on the **Graphic Entity Information** command and then

select any primitive from thecartouche: a window will appear, displaying the layer attribute of the selected primitive.

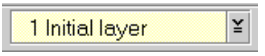
Defining the active layer

Now we want to draw the frame of the drawing. We will place it, with a thickness of 4 and thecolor green, on the same layer of thecartouche (layer 2). Therefore we define the thickness 4, the layer 2 and thecolor green, in such a way that the drawn primitiveswill assume these attributes:

1. Make the define attributemode active, if it isn't already, by selecting the **Define-Modify** button from the **Upper Status Bar** . The define mode is active when you see the writing `Define` on the button.

2. Select the **Layer** button from the **Upper Status bar**.This opens the **Calculator Window**. Enter the value 2, which identifies the layer of the cartouche . Then press **Yes** or the **Return** key to activate the layer2.
3. Select the button from the **Upper Status Bar**. This opens the **Calculator Window**. Enter the value 4, then press either **Yes** or **Return** to activate the thickness 4.
4. Select the **Color** button from the **Upper Status Bar**. This opens the **Color Window**. Select the color green, which is the fourth button on the left of the first line, to activate the color green.
5. All the primitives that have been created from now on will have the layer 2, be green andhave a thickness of 4, until we will change the values of these attributes again.Select the **Rectangle** command from the **Segments Window**. Select a rectangle that includes thewhole drawing, by clicking on a point that is slightly ower on the right in respect to the cartouche and on a point that is slightly higher on the right in respect to the plan (the points to select so that the border is equal to that of the figure are (177,-93) and(-90,94).
6. Redefine, following points 2 and 3the color as blackand thickness as 1.

In BlueCAD there are two other operating modes available for defining layers which, in many situations, can be faster than the one just described:



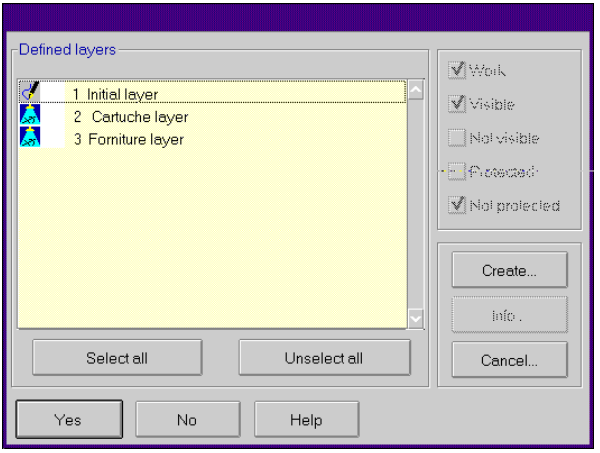
- The **Layer Window** in the **Attribute Zone** of the **Upper Status Bar** shows the current active layer (also called work layer) its number and its description. By selecting the arrow icon on the right of the window, the list of all the existing layers will be shown: selecting a layer from such a list, activates the layer itself.

- In the **Manage Layers Window**, which is accessible with the **Manage Layers** comand, you can activate and deactivate a layer selecting the list from the present layers and clicking on the button **Work Layer:** the check mark on this button indicates that the selected layer is currently active.

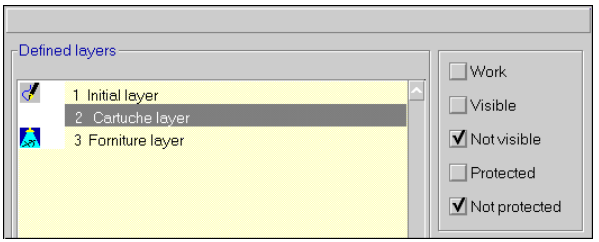
Modifying characteristics and deleting layers

We have previously seen how to create a new layer, giving it also the characteristics of visibility and protection. We can now see how these characteristics can be modified; in particular we want to make layer 2 the Cartouche Layer, non visible and protected, because we haven't drawn on it yet. By doing this, we can concentrate on the parts of the drawing that interest us, therefore speeding up the viewing operations and avoiding accidental deletion or modification of definitive parts of the drawing.

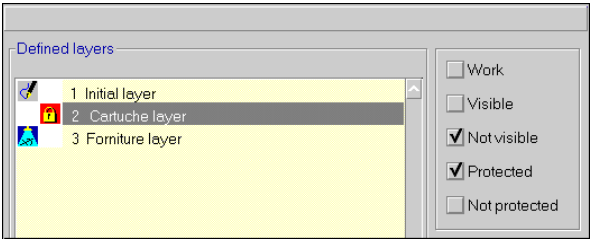
1. Select the **Manage Layers** button. This opens the **Manage Layers Window**.



2. Let's now define the Initial Layer as the work layer. Select this layer from the present list and click the **Work** button. A pen icon next to the name of the layer and a check mark on the **Work Layer** will appear, to indicate that this layer is active.
3. To make the Cartouche Layer non visible select it from the list of the layers and click on the **Non visible** button. A light bulb icon next to the name of the layer disappear, indicating that this layer is no longer visible.



4. To now make the Cartouche layer protected, click the **Protected** button. A lock icon will appear next to the name of the layer, indicating that this layer is now protected This means that the primitives of this layer cannot be deleted nor modified.



5. To apply the modifications select **Yes** or press **Return**.

We have seen how to create and manage the layer characteristics and how to modify and define the layer attribute of the primitives. To delete one or more layers, all you have to do is to select the layers to be deleted from the list present in the **Manage Layers Window** and then click on the **Delete** button; BlueCAD will ask for confirmation before deleting them. To BlueCAD will carry out the deletion of a layer only if the following two conditions have been verified, so guaranteeing a safe operation:

- The layer must not be the work layer. If it is, before carrying out its deletion, define a different layer as the work layer.
- There must not be any primitives on that layer. If there are some, there could be the following two situations:
  - You want to delete the layer (or layers) together with all the primitives of that layer: in this case make all the other layers non visible and then proceed with the deletion of all the primitives of the layer by selecting the **Delete** command with the contextual **Select all** selection; finally delete the layer by clicking on the **Delete** button from the **Manage Layers Window**.

  - You want to delete the layer (or layers) without deleting the primitives of that layer: in this case, make non visible all the layers and then move the primitives from the layer that is to be deleted to another by means of the **Modify Layer** command with the contextual **Select all** selection; finally delete the layer by clicking on the **Delete** button from the **Manage Layers Window**.

Using Blocks

Symbols are graphic primitives that allow you to speed up the drawing process, by means of the reuse of parts of the drawing. Now we want to memorize the door and the window that have been drawn as blocks: in this way they can be stored in a library and reused and shared by other draftsmen. The room will also be furnished with the blocks taken from a library. The blocks management is very simple and powerful. It is made up of a series of basic operations:

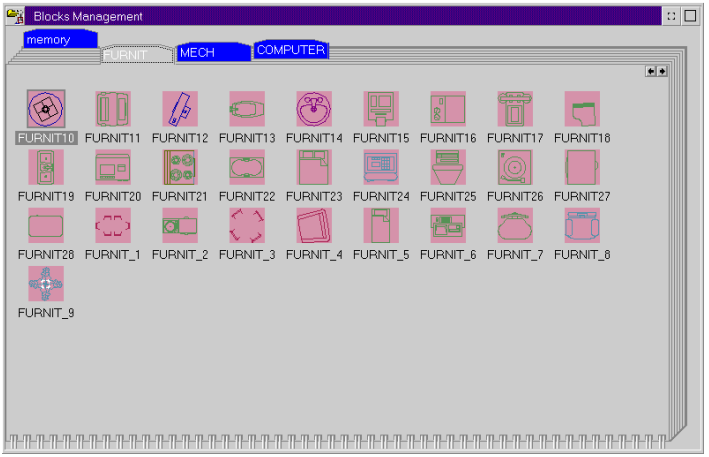
- Creating a library where to store the blocks.
- Creating a block, grouping together the primitives that make it up.
- Memorizing the block in a library of blocks, for its possible subsequent reuse.
- Using a block, taking it from its relative library and positioning it in the drawing.
- Modifing an existent block, creating a different one or splitting it in its component entities.

Important: Every block library physically corresponds to a directory having the same name, which can be found under the BLOCKS directory, in the BlueCAD's installation path. For example the PLAN library correspond to the BLOCKS\PLAN directory. Every block contained in a library physically corresponds to a file having the same name. So, for example, the block Door in the PLAN library corresponds to the DOOR file in the BLOCKS\PLAN directory. Because of this correspondence directory between the library and the symbol and the file, if BlueCAD is installed in a FAT hard disk, the names of the blocks and of the libraries cannot exceed 8 characters. If the installation took place in a HPFS disk then such limitation do not exist.

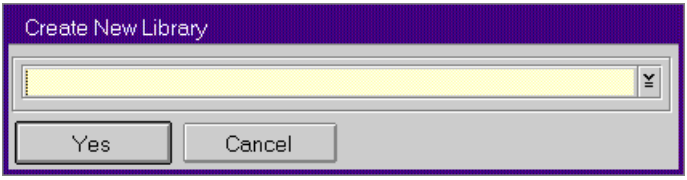
Creating and deleting a library of blocks

The library of blocks are the place where the blocks are stored and from where they are taken to be positioned within the current drawing. We want to now create a library called PLAN where we can store the blocks of the door and window.

1. Click on the **Manage Blocks** button or select the **Blocks** command from the menu **Manage.** This opens the **Manage Blocks Window**. This window looks like a note book made up of many different pages. Each page represents a library of blocks and is identified by a bookmark having the name of the library.



2. Click mouse button 2 when the cursor is in the book mark zone. This will make appear a contextual menu for the creation and the deletion of block libraries: select **New Blocks Library**.
3. The **Create a new library** window will appear. Digit PLAN into the entry field, then press **Yes**: this adds the library to the **Manage Blocks Window** together with the corresponding PLAN bookmark.



Note: The **New Blocks library** command has the ability to make a library of blocks accessible adding it to the **Manage Blocks Window**. If the library is new, that is, it doesn't exist a directory associated to it, then such a directory will be created. Instead if such a library already exists, that is the relative directory exists, then it is only added to the **Blocks Manage Window**. You can see the list of the already existing libraries by clicking on the arrow icon on the right of the **Create new library**;. selecting a library from this list, it will be added to the notebook, if not already present on it.

4. You can leave the **Manage blocks window** either open or closed. In case this window hides part of the graphic area, it can be closed, by selecting once again the **Manage blocks** button. Let's leave this window open as it will be frequently used in the following paragraphs.

The previous procedure can also be used to delete a library, with the only difference that, at point 2 of the above procedure, select the **Delete blocks library** command.

Important: If a library is deleted using the **Delete blocks Library** command its book-mark will disappear from the **Manage Blocks Window**. This library isn't physically erased from the library and neither are the blocks contained in it. Therefore the operation doesn't have any destructive effect, but the library simply isn't available to the user until it isn't made accessible again following the above mentioned procedure. To physically delete a library you must delete its relative directory using the commands of the operating system.

Creating a block

Now let's create the blocks of the door and of the window grouping together the relative primitives.

1. Click on the **Creation** button then select the **Block Creation** command from the **Creation Window**. This command allows you to create a block, specifying the name and origin point and selecting its component primitives.
2. Enter `Door` into the **Strings Window** then press **Yes** or **Return** to confirm the name assigned to the block.
3. Select the external vertex of the door jamb as the block's origin point: click mouse button 2, and then select the **End Point** command from the **Snap points Window**. Then click on the external part of the segment's vertex that represents the door jamb.
4. Now select, all and only, the primitives of the door; to speed up this operation click on mouse button 2 to select the **Select All** command from the **Selection Window**; then click on a point that is situated slightly lower on the left and on a point that is slightly higher on the right in relation to the door.
5. Select the **Data End** command to notify BlueCAD of the end of the selection. Every selected primitive are going to be part of the block until the **Data End** command is carried out: this allows a speedy creation of the block and allows complex blocks to be created.

   Note: Each type of graphic primitive can be selected to be part of a block, including another block. Therefore you can have "nested" blocks, that is blocks inside other blocks.

6. We have now created the door. To verify this, select the **Graphic Entity Information** command and then click on any point of the door: a window will show the information on the block, including the number and the type of primitives that make it up.

7. Repeat the previous procedures, creating the `Window` block using the primitives of one of the windows in the lower walls.
   Note: The name of the block must be unique for a given drawing. If you try to create a block having the name of an already existing block, the message Block already defined will appear in the **Upper Status Bar** and the operation will be unsuccessful.

Memorizing a block in a library

The blocks that were created in the previous paragraphs are automatically memorized in the Memory library: by clicking on the **Memory** book-mark present in the **Manage Blocks window** you can verify that it contains the Door and Window blocks that we have created. The Memory library is a special library, internal to the drawing: each drawing has its own specific Memory library where all the blocks present in the drawing are stored. From from such a library the blocks can be withdrawn and reused within the current drawing. But if we want to share and reuse the Door and Window blocks within other drawings, it is necessary to memorize them in an external library, that everyone can access to. For example, let's memorize them in the Plan library that we have just created.

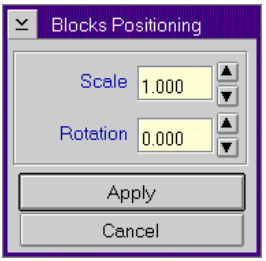
1. Click on the **Plan** book-mark of the **Manage Blocks Window**, to open this library. The library is empty.
2. To memorize the block Door in the Plan library, click on mouse button 2 on any point belonging to the block and then drag and drop it on to the library. During this operation, an icon will indicate that the block has been hooked and that the dragging is taking place.
3. The icon of the block with its name is now in the library, as proof that it has been memorized. Repeat the operation from the previous point, memorizing the block Window, too.
   Nota: The name of a block must be unique within a library data. If you memorize the block in a library where there already is a block having the same name, the already existent block, will be substituted by the new one, after confirmation.

Positioning a block

Once memorized, the blocks can be withdrawn from the librares to be positioned in the current drawing.

Now we want to furnish the room using the furnishing blocks present in the Guide library, positioningthem on layer 3, the Layer of the Furniture.

1. Click on the **Layer Window** button from the **Upper Status bar** to view the existing layers, then click on the Layer of the Furniture list to activate it.
2. Click on the **Guide** book-mark of the **Manage Blocks Window ,** to open this library.
3. We want to position the block of the bed in the drawing. Each block is identified by its name and by an icon which reproduces the block, so that it can be found quickly. In case it cannot be identified, click on mouse button 2 and select the contextual **Search** command to view the list of all the blocks present in the current library:the block Bed will be highlighted in the library by selecting Bed from the list and pressing **Yes**.
4. Drag and drop the block bed in the graphic area. The creation echo supplies a preview of the block.The **Position Blocks Window** automatically opens. This windowsallows the scale and the rotation angle of the block to be controlled



5. Because the dimensions of the beds are too big, enter in 0.5 in the **Scale** field of the **Position Blocks window**, click on the **Apply** button to apply the entered values: the echo of the creation will view the bed with its new settings.

6. The required input is now the positioning point of the block, that is the point where the block will be positioned in relation to its point of origin. To position the bed as it is in the figure select the point (-73,52).
   Nota: The origin point of the block will be identified during the positioning, by the fact that the cursor will hook onto it. When you create a block it is useful to remember that, to make further positionings easy, it is better to choose a significant point as its origin point (for example a symmetry point of the block or a snap point of one of the primitives of the block).

7. Continue to position, with the scale 0.5, the other objects in the room, dragging and dropping them from the library to the graphic area. The Telephone block has been positioned with a rotation angle of 20 degrees.

It could be useful to mention the fact that, if a block has already been positioned once in a drawing, for subsequent positionings it isn't necessary to take it out of a library, but you can drag and drop it directly from and to the graphic area.

Deleting, updating and modifying a block

A block that has been positioned in the drawing can be deleted just like any other primitive, by means of the **Delete** command which is present in the **Modify Window**.

To delete a block from a library click mouse button 2 on the block's icon present in the library, then select the **Delete** contextual command. If you delete a block from the Memory library using this procedure, then all the corresponding blocks in the drawing will be substituted with a label having the name of the block and a cross in the origin point of the block If a block with the same name is then created, then all the labels will be substituted by the new block, keeping the original rotation angles, scales and position points. This allows the substitution and updating of the blocks in a drawing to take place as fast and as efficiently as possible.

The **Break** command from the **Modify Window** allows selecting a primitive made up of other primitives (as in the case of a block, dimension, text, or hatching), to break up in the primitive components. This command is particularly important when you want to modify a block, allowing it to be separated from the primitive that makes it up. To carry out modifications on a block, or to create a new block starting from an already present one, proceed in the folowing way:

1. Position the block, if it isn't already positioned, in the drawing taking it from the library where it is.
2. Carry out the **Break** command; then click on the positioned block.
3. If you want to modify the block, delete it from the Memory library, then recreate it, using the component primitives and carrying out the desired modifications. If you want to instead create a new one making use of the one already present, use the component primitives as a base for its creation, without deleting the block from the Memory library.

Chapter 8: Printing and Plotting

In BlueCAD there are two ways which allow you to obtain a copy on paper of a drawing: printing and plotting.

Printing

When printing during a BlueCAD work session the drawing that you are working on will be printed all the primitives that belong to visible layers will be printed. Printing takes place on the printer set with the operating system, therefore no further print setting is necessary within BlueCAD.

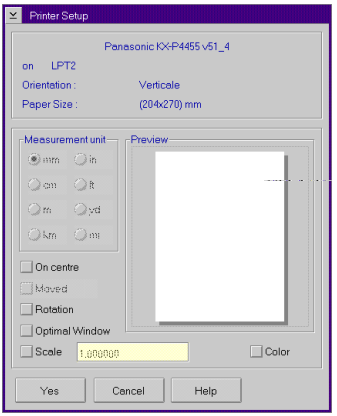
The procedure for printing is the following:

1. Open the drawing that you want to print.
2. Select the Printer Set-up command from the File menu. This opens the **Printer Setup Window**. It is possible to set the print parameters using this window which will be described later on.
3. Select the **Print** command in the **File** menu. This opens a window which asks for confirmation. If confirmation is given printing will begin and a window will open which will show the percentage of the in progress

Printer Setup

The printer setup can be controlled by the **Printer Setup Window** which is opened by the command **Printer Setup** in the **File** menu.

The upper section of the **Printer Setup Window** contains the characteristics of the printer set through the operating system. These characteristics can be modified either by the settings of the printer object on the desktop or by pressing the **Printer** button in this window. If settings are modified in the latter way you have to close and then re-open the **Printer Setup Window**, to make these settings effective



A preview of the printer sheet with a rectangle containing all the visible primitives will be viewed in the window. By means of this preview it is possible to control the drawing as it is printed on the sheet. If there aren't any visible primitives then only the blank sheet will beshown.If a multiple vindow view is active,then the sub-division of the graphic area in the windows will be shown. In this situation you can't activate any print option and the drawing will be printed just as it appears on the screen.

The drawing can be printed in three ways:

- The drawing will be printed just how it is seen on the screen, except for any eventual adjustmentsdue to the different proportions between BlueCADs graphic area and the printer sheet. This way is the only possible ifa multiple window view is active.Otherwise the following options are available

  - ♦ Rotate the drawing by 90 degrees in relation to the sheet, obtaining the same result that you would obtain by modifying the orientation of the sheet from horizontal to vertical (or vice versa) in the printer setup.

  - ♦ Center the drawing on the sheet

  - ♦ Move the drawing on the sheet simply by draggingand dropping it.

  If the drawing is neither centered nor moved it will beprinted making the center of the sheet correspond to thecenter of the graphic area of BlueCAD.

- Printing using the **Optimum Window**. By setting this mode the drawing will be centered on the sheet and the scaling factor, which allows the drawing to occupy all the area of the sheet, will be automatically chosen.In this mode it is possible to choose to rotate the drawing by 90 degrees in relation to the sheet.

- Printing using a **Scaling** factor. In this mode the unit of measurement panel is activated, because it is necessary to specify which unit of measurement corresponds to the adimensional drawing unit. The scaling factor can be set either as a single number or as a formats x/y or x:y. The selected format will be automatically recognized.It is still possible to rotate,center and move the drawing when the scaling option is used.

  Note: If the sheet isn't visible in the preview window, this means that the scaling factor that has been used is too big.Instead, if the recatangle of the primitives is reduced to a point in the preview window,this means that the scaling factor that has been used is too small.

Selecting the **Color** option in the **Printer Setup Window** it is possible to print in black and white or in color. If you activate the color option the entities of the drawing will be printed in the colors that they have been drawn, therefore it is necessary to pay attention to the white entities which will not be visible on the white sheet. If the print is in black and white all the entities will be printed in black.

Note: The printing settings will be memorized when the window is closed. In particular it will be memorized if the drawing has been moved onto the sheet. If this is the case the drawing will be changed and when the **Print Setup Window** will be opened, the movement visualized will be random.

Note: If in the printer driver has the output in postscript format and if the output itself is set on file insead of on the printer port, the print can be used to export the drawing into postscript format.

OS2WARP

Plotting

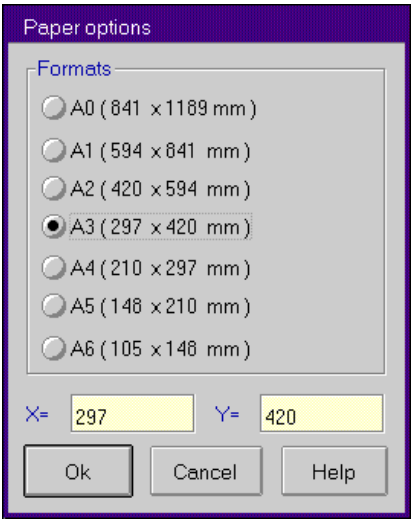
It is possible to print on the plotterusing the BluePLOT program. To start BluePLOT double click on the BluePLOT's icon present in the BlueCAD's folder.

Once BluePLOT has been opened,use the following procedure for plotting:

1. Open the drawing that you want to plot with the command **Read** from the **File** menu of BluePLOT.
2. Set the dimensions of the plotter sheetusing the command **Format** from the **Options** menu. You can skip this point if , as will be described later on, the **Read from Plotter** option has been set.
3. Set the output format, HPGL or Calcomp,andother output options with the command **Output** from the **Options** menu.
4. Set the print options with the command **Print** from the **Options** menu.
5. Carry out the plotting with the command **Plot** on the menu bar.

Format options

Format options are controlled via the **Format Options Window** which is opened selecting the command **Format** in the **Options** menu.

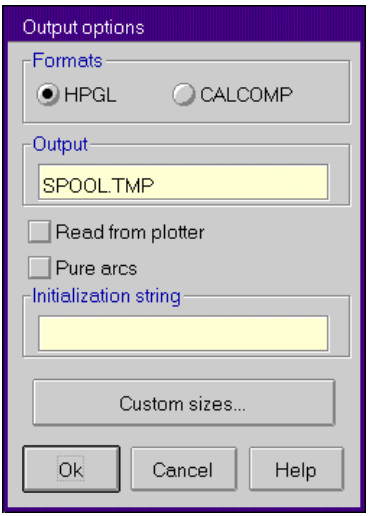


This window allows you to set the dimensions of the plotter sheet. The available dimensions are the ones of the standard formats from A6 to A0. Alternatively you can set a customized format, entering its X and Y values. other different measurements if you want. As mentioned above, if the **Read from Plotter** option has been set, the dimensions of the sheet are read directly from the plotter and the dimensions set in this window will be ignored.

Output options

The output options are controlled with the **Output Options Window** which is opened selecting the command **Output** from the **Options** menu

The output format can be set within this window, and this can be either HPGL or Calcomp. With these two formats it is possible to use all the plotters currently available.



The output can be directed on either a communication port or on to a file. Using Calcomp the plotter can be connected to a serial port but not to a parallel.

It is possible to set the plotting of arcs using the option **Pure Arcs**, otherwise the arcs will be drawn as piecewise linear segments. If this is the case the dimensions of the segments will be chosen in such a way that are small enough to have smooth arcs an unbroken arc on the sheet. The **Pure Arcs** option can speed up the plotting, but sometimes it can create problems.

Selecting the HPGL format some of extra options are available:

- If the output is directed to a communication port it is possible to activate the **Read from Plotter** option to read from the plotter the

dimensions of the sheet. If this is the case then the dimensions set in the **Format Options Window** will be ignored.

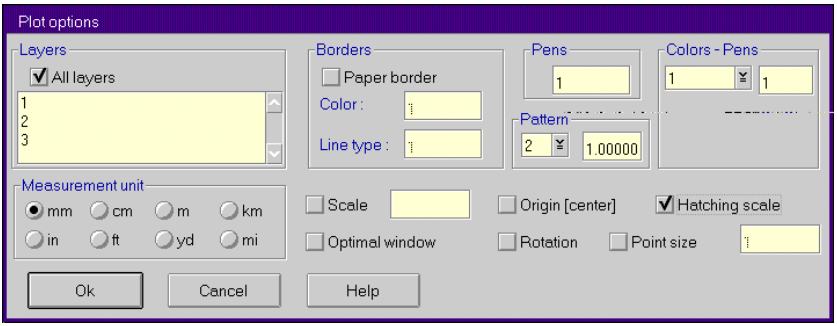
- Some plotter may need an initialization string to work properly. Consult the manual of your plotter to find out if such a string is essential and to find out of which characters it is made up. If this should be the case, digit it in the **Initialization string** filed.

- With some plotters the dimensions of the drawing that has been plotted don't result correct. In this case you must plot a drawing having known dimensions, unit of measurement and scaling factor, for example a square with side 10, in centimeters and with scaling factor 1. Then you must measure the dimensions of the figure on the sheet after plotting. Finally, you must feed all these measurements into the **Custom Sizes Window**, which will call the **Output Options Window** BluePLOT will automatically adjust the dimensions of the entities that have been plotted so that the mistake will be corrected. This setting when necessary, is only used once, because it will be memorized at the end of BluePLOT's work, just as all the other settings.

Note: If the output is directed on to a file, the plotting can be used to transfer the drawing in format HPGL or Calcomp, according to the output format that has been set

Print Options

The print options are all controlled via the **Print Options Window** which is opened by the command **Print** from the BluePLOT's **Options** menu.



First of all you must select if all the layers are to be plotted or if only some of them and which ones.

Then you must decide whether the drawing is to be printed using the view that it has been saved with or by using a defined scaling factor or in an optimum scale. If you use a definite scaling factor you must also set a real unit of measuremnet that corresponds to the (adimensional) drawing unit. The scaling factor can be set either as a single number , or as a format x/y or x:y. If you use the optimum scale BlueCAD automatically calculates and applies the optimum scale for the drawing to plot. If the drawing has been saved with a multiple window view and the scaling option is used, only one window will be plotted.

You can set a rotation of the drawing of 90 degrees in relation to the sheet. This is possible in a drawing with a multiple windows view only if the scaling option is also active (in fact, as we have seen, with the the scaling option only one window will be plotted).

If the output format is HPGL it is possible to decide if the drawing will be plotted in the middle of the sheet or starting from the bottom left hand corner.

You can plot a drawing frame, controlling its color and line type, or you can plot without a frame.

You can establish the number of plotter pens and the association color-pen: BluePLOT associates to every BlueCAD's drawing color a plotter pen. Therefore you can define for every color the pen with which it must be plotted. When printing on the plotter the thickness is unimportant the thickness is controlled together with the color, by choosing the pen. You can set the line type by setting the length of the pattern line for every non-continuos line type.

Finally you can control if the hatchings must be scaled in relation to the perimeters that contain them. If this is the case, the hatchings are plotted exactly as they are drawn on the screen by BlueCAD. Otherwise, the step with which they will be plotted, will be the real one, taking into account the defined unit of measurement. It is also possible to define the dimensions that plotted points must have or let them have an arbitrary dimension.

Note: All the options are memorized when BluePLOT is closed and will be the active ones when BluePLOT is re-opened.

Third Part- Advanced drawing tecniques

In this Third Part will be explained the use of the dimensions, and also the information exchange techniques between BlueCAD and other software products. Furthermore, we explain how to use the REXX (OS/2 Warp), or C\C++ (Windows) programming language to customize BlueCAD, creating new commands. With this third and last part, the tools for using BlueCAD in a professional way are supplied. In addition, it gives a vision of BlueCAD as an integration tool in a complex work area, in which, passing from drawing to project, it is necessary to exchange information among different products, and also to use the programming characteristics to make it a system as open and flexible as possible.

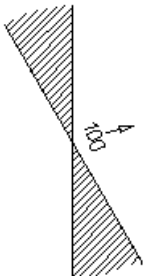
Chapter 9. TheDimensions

To be able to build an object that has been drawn with BlueCAD , you must know its dimensions. These are supplied by the dimension entities. BlueCAD supplies a group of dimension functions that respect the UNI-ISO standards. They are associative and allow the tolerances to be controlled. So once an object has been drawn the dimensions can be viewed by dimensioning it.

Using the dimension

For a good dimensioning the dimension entities must be put on the drawing in such a way so that under no circumstance must the user have to do any arithmetic operations. The dimensions are then rationally disributed, in such a way to make the most of the space available in the drawing. Therefore you must avoid the crowding of dimensions on only one part of the drawing and the use of not necessary dimensions.

As illustrated in the figure, dimension lines in a sector of 30 degrees must not be placed. This can be done only in extreme cases of need.



BlueCAD's dimension is associative, that is, it remembers the graphic entity selected for its creation. When this entity is transformed (moved or stretched), the dimension is automatically modified to fit in with the new geometry. In the case of the modification of the entity the dimension will be

also modified or eliminated according to the dimension of the modified entity makes sense or not.

The dimension text will be automatically oriented by BlueCAD so that it can be read straight in the direction of the arrow in the figure.

Creating a dimension

BlueCAD's dimensions are of different types , according to what they have to measure. Click on the **Dimension** button to use the dimensions creation commands, this opens the **Dimension Window**. The commands are:

- **Dimension 2 Points** this dimensions the distance between any two points.

- **Dimension Segment** this dimensions the length of a segment, that is the distance between its two end points.

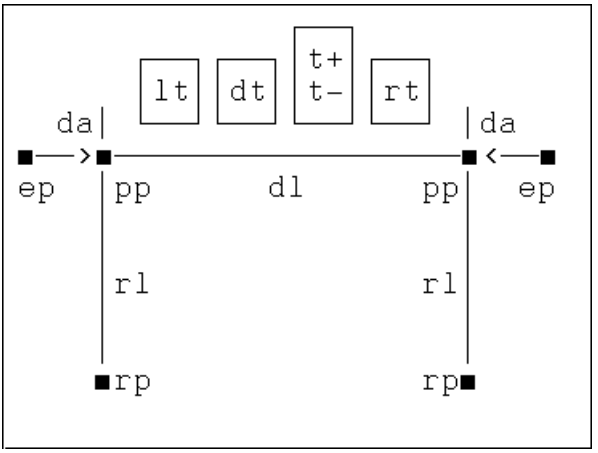
- **Dimension Angle**: this dimensions the angle between two segments.

- **Dimension Diameter**: this dimensions the diameter of a circle or of an arc of a circle.

- **Dimension Radius** this dimensions the radius of a circle or of an arc of a circle.

- **Dimension Parallel Segments:** this dimensions the distance between two parallel segments. It controlls the condition of parallelism between two segments.

The following figure schematically views a typical dimension:



abbr.	**description**
da	arrow
dl	line dimension
rl	reference line
ep	extension point
pp	projection point
pr	reference point
t+	upper tolerance
t-	lower tolerance
rt	right text
dt	dimension text
lt	left text

The dimension creation commands require either two points or the selection of the entity to be dimensioned and the positioning point of the dimension text. The dimension text shows you the value of the dimension. This value is in the (adimensional) drawing unit if referred to a distance, in 60ths of degrees if referred to an angle. This value: if it is a distance in the (adimensional) unit of measurement of the drawing, if it is an angle measured in 60ths of degrees. If in the **General** page of the **Settings Window** the **Optimum Dimension** is active, then the dimension text will be

positioned in the optimum position, in relation to its selected positioning point.

Modifying a dimension

You can carry out the following modifying operations:

- **Modify dimension**: this requires the selection of a dimension and of the point where this dimension text is to be repositioned. In this case the **Optimum dimension** option influences the new position of the dimension text.

- **Detach dimension**: this eliminates the association between the selected dimension and the primitive to which it refers: subsequent modifications of this primitive will not have any influence on the dimension.

- **Modify dimension text**: this requires the selection of a dimension and of the new text that is to be assigned to the dimension. Modifying the dimension of a text allows you to substitute the dimension text with any other text. This can be useful for dimensioning details that are out of scale, or for giving to a dimension a code instead of the value. It can also be useful for inserting the text on the sides of the value of the dimension: the position of the optional # char in the dimension text, identifies where to insert in the text string the value of the dimension.

Dimensions attributes

The tolerances are values associated to all the dimensions, except the angle dimensions, which indicate the maximum difference possibile between the real dimension and the nominal dimension of an object. There are two tolerances, the upper tolerance and the lower tolerance , and they are represented, when they are present, as two texts, of hight equal to 0.45 the height of the dimension text.

The two texts of the tolerances are positioned to the right of the dimension text. If you define the value of one tolerance automatically also the other one will be defined with a null value.

The tolerance commands are among the dimension attributes commands.

Dimensions attributes

The commands for dimension attributes are

- **Arrow type**: this defines or modifies the shape of the arrow of the selected dimension. From the **Arrow Type Window** select the arrow type you want

- **Dimension Gap**: This defines or modifies the gap of the dimension. If you set a positive dimension gap you define the distance between the reference points and the reference lines of the dimension. If you set a negative dimension gap its absolute value defines the length of the reference lines, which will start from the dimension line and will be separated by the reference points. This attribute is useful for separating the reference lines from the primitives dimensioned.

- **Decimals Number**: this defines or modifies the decimals number with which the value of the dimension is viewed (from 1 to 4). The decimals number of the tolerances is fixed at three figures after the comma. It is also possible to specify a negative number: in this case the number of decimals is its absolute value, with the difference that the non significant zeros are eliminated. The non significant zeros of the tolerances are also eliminated.

- **Upper tolerance**: this defines or modifies the value of the upper tolerances of the dimensions.

- **Lower tolerance**: this defines or modifies the value of the lower tolerances of the dimensions.

- **Delete tolerances**: this nulls the values of the tolerances.

Note: The dimension text attributes are defined and modified (except the **Origin text)** by using the text attributes commands.

Chapter 10. Information Exchange

BlueCAD puts the following tools at your disposal for exchanging of information with other programs:

- Reading and writing of drawings in DXF format.
- Reading of images in bitmap formats.
- Copy of the drawing, all or in part, on the system clipboard in bitmap or metafile format.
- Writing of the drawing in postscript format through the print command.
- Writing of the drawing in HPGL format through the BluePLOT program.

Import and export of drawings in DXF format

The DXF format enables the exchange of drawings with CAD programs and the most widespread graphic programs.

To read a drawing in DXF format, select the **Import Dxf** command from the **File** menu. This opens a window for the selection of the file name to be read. The DXF file is read without deleting the current drawing. It is therefore merged to the current drawing, not substituted by it as the **Read** command does.

To write a drawing in DXF format, select the **Export Dxf** command from the **File** menu. This opens a window for the selection of the file name to be written.

Importing bitmaps

BlueCAD enables the insertion of images as a background of the drawing. The images can be in one of the following formats:

ext.	format
BMP	OS/2 and Windows bitmap
TIF	Microsoft/Aldus Tagged
GIF	CompuServe Graphics Interchange
PCX	ZSoft PC Paintbrush Image
TGA	Truevision Targa/Vista bitmap
IAX	IBM Image Access eXecutive
XBM	X Windows bitmap
IMG	GEM Raster

To insert a bitmap:

1. Click on the **Creation** button, opening the **Creation Window**.

2. Select the **Read Bitmap** command.
3. Select the bitmap file using the selection window, which is automatically opened by BlueCAD.
4. Select a point on the graphic area: the image will be inserted with its left corner on this point.

To scale an inserted bitmap:

1. Click on the **Modify** button, opening the **Modify Window**.
2. Select the **Scale bitmap** command.
3. Select two points of the bitmap.

4. Set the distance that you desire between these two points. The distance is inserted in the arbitrary drawing unit of dimensioning. The bitmap will be scaled in a way that the distance between the two selected points is exactly the distance set.

To delete a bitmap:

1. Click on the **Modify** button, opening the **Modify Window**.

2. Select the **Delete bitmap** command.
3. Select the bitmap that you wish to delete.

The visibility of the bitmaps can be controlled through the **Bitmap Viewing** setting in the **Settings-General Page Window**. It is useful to deactivate the viewing of the bitmaps to speed up the redrawing operations. If the bitmap viewing is deactivated, the bitmaps are not printed.

Note: the bitmaps can be printed with the print command, but they cannot be plotted with the BluePLOT program.

Copy of the drawing on the system clipboard

You can copy the drawing, all or in part, on the system clipboard. The copy is possible either in bitmap format or in metafile format. The drawing copied on the clipboard can be read by all programs that have access to the system clipboard. The most convenient format to use depends on the program that you intend to use. For example, you can copy a drawing in a specific bitmap format to then insert it in a document of a word processor to produce technical documentation.

To copy on the clipboard in bitmap format:

1. Select the **Clipboard, Copy image** command from the **Edit** menu.
2. Select the zone of the drawing to be copied on the clipboard.

To copy on the clipboard in metafile format:

1. Select the **Clipboard, Copy entity** command from the **Edit** menu.
2. Select the primitives to be copied on the clipboard. Click on **Data End** to end the selection of primitives.

Note: The metafile produced by BlueCAD like this can be saved rather than . You can do this through the option present in the **Settings-Metafile Page**, which enable you to specify the name of the file to which BlueCAD will save the metafile.

Exporting the drawing in PostScript format

As we have seen in Chapter 8, if the printer driver has a postscript format output and the output is set on a file rather than on the port of the printer, the print exports the visible entities of the drawing in postscript format on this file.

Exporting the drawing in HPGL format

As we have seen in Chapter 8, if the BluePLOT's output is directed on a file and the HPGL output format is set, the plotting exports the drawing in HPGL format on this a file. Using BluePLOT, it is also possible to export the drawing in Calcomp format.

Chapter 11. The Macros

BlueCAD gives you the possiblity of creating macro commands in REXX (OS/2 Warp), or C\C++ (Windows) language to be used like additional commands. The macros are the basic tools for customizing the program, adding new functions to it.

Using the macros

The management and the use of macros comes according to the following procedure:

1. Through **Manage Macros** the macros can be inserted and eliminated in the **Macro Window** and created, tested, modified.
2. The inserted macros can be carried out through the **Macro Window** buttons

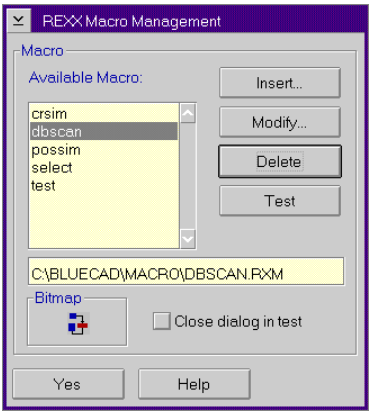
The macros are identified internally by a name, which a bitmap and a file are associated too. The macro and the bitmap are memorized in files having the macro name and exstension respectively **.rxm** and **.bmp**. The bitmap is associated to the relative macro button in the **Macro Window**. The file contains the procedure that is carried out when the macro is executed.

The REXX language is put at your disposal by the operating system to be able to write procedures normally used as command files (see the documentation coming with the operating system). BlueCAD macros are REXX procedures that can access all the functions available through the operatinge system for the procedures written in this language In addition, BlueCAD offers a set of functions that allow you to access to the BlueCAD's

Note: At the end of the work session, BlueCAD memorizes the situation of the inserted macros to then restore it in the next restarting of the program.

Macro Management

Select the **Macro** command in the **Manage** menu. This opens the **Manage Macro Window**.



In this window a list of available macros is presented, that is of those already inserted in this or in previous work sessions. If a macro from the list is selected, the name of the file that contains the procedure and the associated bitmap appear. You can:

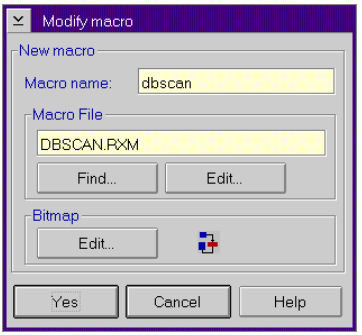
- create a new macro.
- modify the selected macro.
- eliminate the selected macro.
- test the selected macro.

Creation and modify macro

To create or modify a macro, open the **Insert Macro Window** by clicking on the **Insert** button in the **Manage Macro Window**. In this window you can insert or modify the name of the macro and the name of the file containing the macro procedure. From this window you can launch the

system editor to create or modify the file containing the procedure. You can also launch the system icon editor to modify the bitmap associated by default to the macro.

Note: When the editor is launched, its window could appear behind the BlueCAD window and is therefore not immediately visible. In this case press **CTRL+ESC**: the **Windows List** appears in which the editor is listed under the name BlueCAD, with the name BlueCAD.



Note: There are some sample macros available. From the **Manage Macro Window** click on **Insert** to open the **Insert Macro Window.** Click the **Find** pushbutton of the **File Macro** frame to see the list of available macros.

Elimination of a macro

If the macro selected in the list is eliminated, this macro doesn't appear anymore in the **Macro Window**. This operation does not have any consequence on the macro file and on the bitmap file associated that are not deleted. Thus, this macro could easily be reinserted successively.

Test of a macro

When testing a macro, the macro is carried out in the same way as it was executed from the **Macro Window**, with the consequent modifications in the current drawing. You can choose to close the **Manage Macro Window** during the test.

Macro Window

Click on the **Macro** button from the **Vertical Toolbar**, opening the **Macro Window**.

You can execute the macros inserted previously with the **Manage Macro Window**. To execute a macro just select its icon from the **Macro Window**.

Glossary

2D
Geometry is represented within the CAD database as x and y coordinates. The CRT screen is simply a model of the draftsman's sheet of paper. The geometry placed on the screen is the same as the lines drawn on a sheet of paper. Each view must be independently drawn.

2D½
In a CAD system of this type the geometry is represented by means of points which are copies of the x, y coordinates. However, facilities are provided to project the geometry in 2D creating the visual illusion of a three-dimensional model or drawing.

3D
Geometry is represented within the CAD database with x,y and z coordinates. A three-dimensional model is maintained by software. The CRT screen represents one or more views of the model as viewed from different user-defined orientations.

3D model
A model which has width, length and height.

3D: projection in 2D
Some CAD systems have the possibility to create a two-dimensional representation of a 3D object by extruding the drawing and performing the same manipulations in the computer that would be performed on paper to generate perspectives and axiometrics.

3D Wireframe
A representaion of a real or proposed physical object where the representation consists of line segments. This is similiar to creating a figure with sticks.

A

Absolute coordinates
Coordinates in relation to the origin point (a point with null co-ordinates).

ANSI
American National Standards Institute. One of the most important and reliable institute in the world as regards to standardization directives. It is also involoved in the field of software.

Application
A program that performs specific tasks such as word processing, spreadsheet or CAD. A CAD application may also have application-programming tools within the program for an advanced use. Macros are an example of applications programming tool.

Arc (of a circle)
Part of a circle included between two points (end points).

Area
The measurement of a closed surface.

ASCII
American Standard Code for Information Interchange. A standard format for description of letters, numbers and special characters. Each character coincides with the processor with a code of 8 binary numbers.

ASCII TEXT
Text that only uses 7 bits per character following the ANSI (SEE: ANSI) standard. Therefore only 128 letters are representable. This format guarantees the portability of files from one computer.

Associative dimensioning
CAD-feature for updating the dimensions automatically when the primitives to which they are associated are modified.

Associativity
A relationship between entities. Usually one entity is defined as the parent entity and the associated entities are called children entities.

Attributes
Facts. Words or phrases which serve to limit, detail, describe or supplement the meaning of the feature to which it is attached. Color and thickness are examples of line attributes. Text attributes are font, slant, height, width, spacing.

B

Back up

The making of duplicate files for use in case of damage to the originals or for archival purposes.

Baricenter

The center of mass of a system made up of a finite number of points of equal mass.

Batch file

A file containing several commands, which are executed one at a time, typically in an unattended mode. Enables the user to repeat a given sequence of commands several times without having to rewrite them.

Bezier curve

A smooth curve consisting of a series of four control points that guide or influence the direction of the curve. The curve doesn't necessarily pass through the control points, although two of the points determine the direction of the curve and the other two are endpoints. Bezier curves are used extensively in the design of surfaces such as those of car bodies.

Binary

The arithmetic basis for calculations in all computers. It is a two-digit numbering system consisting of the digits 0 and 1, in contrast to the ten digit decimal system.

Bit

Binary digit. Smallest possible unit of information, reprsented as either 0 or a 1. Eight bits equal one byte.

Bitmap

It indicates the group of values that describe, in an on screen output device, the assumed value (color) of every element of image. For example, if a screen has 600*800 of such elements, a group of 4.8E5 values are necessary.

Bitmap, Format

It is the format with which an image is saved, memorizing the position and the color of the pixel on the screen. For example, let's suppose that the image is made up of a simple segment, the position of all the elements that are part of the image will be memorized. This format takes up more memory but, above all, it does not contain any geometrical information, therefore it isn't possible to ask the ask the calculator the length of the segment.

Block (Symbol)

Graphic primitive made up of a group of more graphic entities, to which a name is given. You can proceed to copy the block in other areas of the drawing simplyby choosing the block from the block library.

Boundary

One of three basic methods for representing solids in the 3D CAD systems. It contains the description of every face, corner and vertex. References can establish relationships between entities.

Byte

A group of eight bits. Usually expressed in thousands or millions.

C

CAD

Computer-Aided Design and Drafting. The combination of computer hardware and software that is used to automate the manual procedures of design. Now used interchangeably with CADD(Computer-Aided Design and Drafting). Such activity can be carried out by various input devices (keyboard, mouse, graphic board), while it is also possible to introduce purely numeric data delegating the elaborator with the true stages of drawing. Another possibility is to, having an image available, translate it onto magnetic supports with the correct devices (videocamera, scanner) and then elaborating it.

CAE

Computer-Aided Design/Computer Aided Engineering. Any system or software whose primary function is to use computer-generated geometry, graphics, associative data and text-to-design and maintain models of the real world. CAD/CAE systems normally include central processors, software, networks and peripherals.

CAM

Computer-Aided Manufacturing or Computer-Aided Mapping.It indicates a further process of automation and of control in the realization of projects. In other words, the drawings and CAD information will be enriched with information regarding the productive process, generating information that is sent directly to the machine tools. All of this involves the use of a database, from which the user can generate product details and can build sequences of numeric control for controlling machinery.

Cartesian co-ordinates (Absolute Rectangular)

A system of coordinates that consists of a pair of axes of right-angled coordinates with the vertex in a point called origin. The position of a point in the reference system is specified by means of a pair of real numbers, a distance of the point in relation to the origin along the axes.

CD-ROM

Compact Disc, Read Only Memory. A portable storage disk read by optical means, instead of electromagnetic. Similiar to the CDs used by stereos.

Chamfer

A proceedure which creates a beveled edge between two intersecting lines. In case of two segments then a slanted segment is inserted, in case of two planes, then a slanted plane is inserted.

Check mark

A symbol that indicates thata selection is active, used in the dialog windowsand in the chek mark boxes.

Circle
A closed line made up of all the points of the surface that have the same distance (radius) from a point (center).

Circular copy (Array circular)
In this case the generation of copies comes at equal distances, or at right angles, along the arc of a circle.

Click
Term to indicate the non-repeated pressure put on the button of the mouse.

Clipboard
An area of the memory which contains temporary data whichis exported from one program to another.

Command
In general, a computer instruction.

Computer
A device that receives, processes and presents data. The two types are analogical and digital. Also known as a computerized machine.

CONFIG.SYS
An operating system file where the system where the system configuration is addressed. It indicates to the operating system, in the starting phase, which type of device is available (for example, printer, video, memory etc).

Consecutive primitive
Two grahic primitives are consecutive when they have a common end point.

Context sensitive on-line help
Readily available on-screen information related to the current command or function which you are working with.

Coordinate
A group of numbers that localize a point in space (in 3D) or on the plane (in 2D).

Co-processor

A specialised processor with minor abilities as compared to the central processor. It assists the main computer in performing certain types of instructions, thereby speeding up the processing.

CPU (Central Processing Unit)

Central Processing Unit. The brains of a computer.

D

Database

A collection of interrelated data stored together to serve one or more applications. The characteristic of every database management program (called database engine) is the speed with which it consults enormous amounts of data, in modifying it and in replying to precise requests of information (queries) from the user..

Data Type

Defines the type of data that will be contained in a variable, whether it is a letter, a whole number, a number in a scientific format.

Decimal degree

A unit of angular measurement where the angle rotation is always of 360 degrees, but the fractions of degrees are expressed in decimal fractions and instead of minutes and seconds. Example:125.541° .

Default

The function, value, etc., which is chosen if no other choice is made by the operator.

Degree of a polynomial

Maximum potence present in a polynomial. Example: in $(3x^4 + 5x^3 + 7x^2 + 8x + 5)$ the degree is 4. The degree of a spline function is important in the definition of a spline function because it modifies the curve.

Dialog Box

A portion of the screen intended for the user to select from a set of options or fill-in with information.

Dimensioning

Dimension is the text that every geometric graphic entity is associated to. Graphic elements of dimensioning are the line of dimension, the reference line, the dimension text. The reference lines are two short segments that highlight points between which the measuring takes place, the dimension line is a segment traced between the reference lines parallel to the measured dimensions, while the dimension text relates the measurements of the dimension.

Directory

One of the branch, in a tree like structure, in which the user can sub-divide a unit of mass memory (hard disk, floppy disk etc.) so as to organize the information better, e.g. the gathering of files and therefore their rapid search. A tree like structure means that every directory can branch off into a sub-directory, for example containing files.

Display Drive

SEE Video Adapter

DOS

Abbreviation for Disk Operating System. One of the most used operating systems, produced in various versions by various informatic houses. PC DOS is produced by IBM.

Double Click

Indicates the double quick pressure of the same mouse button. Usually such an action starts a command different from the one associated with the click.

Drag and Drop

The following operation is meant by this expression: move mouse pointer on an object, hold down mouse button 2, move it towards the desired position (dragging the object) and then release mouse button 2 to drop the object.

Drawing

The paper rapresentation (or the file used to create it) of a model used to communicate the design during all stages from conception through manufacturing.

Drive
An electronic card for the control and functioning of the communication protocol between the calculator and its devices.

DXF
Drawing Exchange Format.One of the most common formats for drawing data interchange, defined by AutoCAD.

E

EGA
Enhanced Graphics Adapter. An industry standard of control language and resolution. Medium resolution display (640 x 350 pixels). This standard was set by IBM.

Background processing
An activity of ordinary maintenance that the processor carries out independently from the activity that the user is currently doing.

Entity
A geometric primitive or building block, used to create drawings or models. It is an object that is defined and can be separated from the rest of the drawing and selected on the screen.

Export
Transfering of data from one system to another. It usually implies a conversion activity between different modes of codifying information.

F

FAT (file allocation table)
Management system of the file that uses the FILE ALLOCATION TABLE. Used by DOS and by OS/2, it has the limit of 8 letters for the name of the file and of 3 for the extension.

Fast selection (Shortcut, Function Key)

One or more combinations of keys used to send a more complex command to the computer(as a subroutine, a macro or a utility program), or to avoid having to give up the keyboard during the work session for the mouse.

File

A collection of related information, usually stored on disk. A file can contain data, programs, or both. A named block of text, alphanumeric data, program instructions, commands or graphic data. This data can be interpreted by programs. It may reside in secondary storage, but it must be transferred to memory for manipulation and use.

Fillet

Method of automatically inserting a graphic entity between another two: an arc of a circle if it is between two segments, a cylindrical sector if it is between two floors.

Floating points

Method for the numerical representation in an elaborator. Each number is converted in the form (mantissa*10). Such a method allows numbers also to be treated, with decimal parts, that go over the limit of the machine for integers.

Floppy disk

A portable, flexible mass-storage disk typically with less capacity than hard disks.

Font

A style (shape) defined for a whole series of characters.

Format

A group of conventions and rules to be respected in the codification of data and the writing of files, so that the machine works correctly.

G

Graphic attribute

Any viewable characteristic of an entity as displayed on screen. For example color, line width, font etc.

Graphic board (Digitizer, Digitalizer)

A digitalizer is made up of a flat surface on which a pointing device is slid. It is a device that interacts with the pointer and recognizes it s position, making it correspond to a specific point on the screen in an direct way. All this allows the screen to be used just as if it were a sheet of paper.

Graphic co-processor

Similiar to the mathematic co-processor, but it supports the CPU in carrying out certain drawing activities.

Graphic primitive

SEE: Entity

Grid

A matrix of right angled lines present on the screen but not part of the drawing, auxiliary in the projection activity. It is assimilated to millimetered paper in the drawing manual.

GUI (Graphic User Interface)

Part of the operating system that enables interaction between the user and the computer, generally without requiring text based commands to be memorized and typed. It has substituted the "on-line commands". Easy to understand, you interact using the mouse, intrinsically slower than the keyboard.

H

Hard disk

Mass storage device which is thin circular and covered with magnetic material.

Hardware

Term used to indicate the physical components of the computer processing system, including mechanical, magnetic and electronic devices.

Hatching (Crosshatch)

A regular pattern of lines and symbols that fills a bounded area. Hatchings are used, for example, in mechanical drawing to indicate the different types of work, in digital mapping (the production of geographical map assisted by the database) to indicate different surfaces (arable, arboreal, deciduous woods,pasture..)

HPFS (high performance file system)

File system management used by OS/2. The names of the file used with HPFS can be upto 254 letters.

I

Icon

In a graphic interface operating system, an icon is an image, usually of reduced dimensions, on which the cursor of the mouse can be positioned. The icon is associated to a command or a function that can be executed by clicking on it.

Import

To receive data from an outside source. Usually itimplies the activity of conversion between different modes of codifying the information.

Integer

A positive or negative number without any fractions.

Interactive

A mode of operation in which alternating entries between an user and the system take place in a manner similar to a dialogue between two people

Interface

In general, it indicates the point of contact, logical or physical, between two entities.

Intersection Cleanup

In CAD programs where parallel lines form intersections, or overlapping line segments, those intersections are automatically cleaned up with a single command.

ISO
International Standard Organization.

K

K (kilo)
Prefix for a unit of measurement that indicates the multiplication by 1000. For example: 2 KCalorie equals 2000 Calories.In the case of computer bytes, 1Kbyte stands for 1024, and that is 2^{10} bytes.

Keyboard
The most common kind ofinput device. Pressing a key corresponds to sendingto the processor the 8 binary numbers that codify it, usually according to the standard ASCII.

L

LAN (Local Area Network)
A computer network geographically limited, assisted by physical support dedicated, that is, without relying on interconnection via modem by telephone network.

Layer (Level)
In the CAD systems it is possible to organize the drawing in different layers. Each layer contains graphic information is given according to a sub-division defined by the user. For example, in a building project you could designate the different floors of the building being projected to different layers, keeping separate the electric plant and the plumbing structure or the conditioning plant. It is possible to work on single layers making them invisible, so as to highlight those that are of interest at the moment, or to print them selectively.

Level
SEE: layer

Library of blocks
An area reserved in the memory where the user of the CAD system can copy and gather the blocks that are most frequently and which can be easily accessed

Linear Copy (Array linear)

A method for generating multiple copies. The objects lie equispaced on a segment.

Line type

An attribute that specifies the appearance of a line. Eg: continuous, dashed, points and line.

M

M (mega)

Prefix for a unit of measurement, it indicates the multiplication 10^6. Eg.: 1 Mmetre equals 1.000.000 meters. When it prefixes bytes, however, it indicates the multiplication by 1048576.

Macro (Macroinstruction)

Given a sequence of commands that the user has to frequently give, it is useful to memorize the whole sequence and to associate a name to it. Recalling it, brings about the execution. Use of macros can improve greatly the productivity.

Mathematic co-processor

Assits the microprocessor in carrying out floating number operations.

Message Area

An area of the display for prompts, error messages and the like.

Minidisk

A magnetic disk closed in a protective case, used to memorize information.

Mirroring

A command that allows the automatic creation of the symmetrical primitive relative to an axes or, in the case of the 3D system, in relation to a plane.

Module

A program parts of an integrated package of programs.

Mouse

Pointing device with which it is possible to move the cursor on the screen. It has two or three buttons from which you can give commands.

Multitasking
Executing several tasks(usually programs) by one or more users at the same time. Requires a complex operating system specifically designed to handle several tasks at once, and enough RAM to allow access by several tasks.

Multithreading
A thread is an activity that consists of a sequence of instructions. In a computer the multithreading indicates the capacity of the operating system to carry out at the same time more threads.

Multiuser
An operating system that allows more than one user to make use of its resources at the same time under its supervision so as to avoid interference and conflicts.

N

Network
A number of computer all connected together and communicating with one other.

Normal (Perpendicular)
Said of each of the two lines(or planes) that, intersecting, form four equal angles.

O

Operating System
It is the most important part of the software, without which the computer would be useless.The operating system, in fact operates on the accessibility of the computer and manages all of its interactions with the user, with its programs and its devices.

OS/2 Warp
Standard IBM 32 bit, mutithreading and multitasking operating system for PC.

P

Pan (Panoramic)
Command relative to the viewing of a drawing. Consists in the translation of the drawing in relation to the screen along a preferred direction, so that different portions can be viewed.

Parallel port
An output interface with the computer through which all eight bits of a data byte are sent at one time, on parallel. SEE Serial Port.

Piecewise Linear
A line make up of consecutive segments.

Pixel
Stands for PIcture ELement and indicates each element of the rectangular matrix which the screen is divided into.

Platform
Term which indicates the combination of Processor-Devices-Operating System.

Plot
Printing operation on the plotter.

Plotter
Printing device. The plotters can be divided into two categories: those that work by vectors, which draw lines, and those that work by matrix, which draw points. They can use three different printing technologies: The drum type ones are vectorial. They can produce complex and big color drawings; their plotting pens slide horizontally on a drum which unrolls the sheet. Table plotters mount a pen that slides along an axis, while the whole axis can move itself along the right angle directions. Usually they are of reduced dimensions. Finally, the elctrostatic plotters make the points stick to the area of the sheet that have been electrostatically charged: such devices are not the most effective but they are fast.

Polygon
A closed broken line.

Polar Coordinates
A reference system consisting in a definite origin point and in a semi-straight line, to which the origin belongs. The position of any point on the surface is identified by an angle and a distance. The point is found on the semi-straight line with the vertex in the origin, forming the angle.

Printer

There are two types of printer. The dot printer makes the ink stick to the paper, leaving in this way its trace: these are the cheapest, noisier and of a lower quality. The jet ink printer sprays little drops of ink onto the paper: these are silent, fast and produce prints of good quality. The laser printer electrostatically charges areas of the sheet on to which it then adheres black toner: these are fast and silent.The printer uses a heating process which stamps on to the paper a wax based ink: this allowscolor printing.

Program

Sequence of instructions in a language that is comprehensible to the computer.

Prompt

A message which asks you to perform some action, usually entering information.

Q

R

RAM

Random Access Memory. Volatile memory for quick access destined to receive data from the user about which the elaborator must carry out its own activity.

Raster, format

Almost identical to the bitmap format, usually indicates that the file comes from the activity of the scanner.

Rectilinear Copy (Array rectilinear)

A command for generating rapidly a multiple copy of a graphic object in relation to a rectangular path (the title could be confusing).It is realized by means of a multiple copy of the object along a straight line at equal distances, therefore with a replica of such a series in a right angled direction.

Redrawing
A command that updates all that appears in the current work view taking into account the latest changes made to the drawing.

Reference lines
SEE: Dimensioning.

Regular polygon
It is a polygon whit its sides of same length and the angles of the same value. For every regular polygon there is an inscribed circle, tangent to the sides in their mid points, and a circumscribed circle, passing through its vertices.

Relative Coordinate
A coordinate relative to a given reference point.

REXX
Programming language for writing simple and advanced applications to use whitin OS/2.

Resolution
It indicates the number of pixels that the screen is divided into. It is defined *high* for values between 1024*768 and1280*800,*very high* for values between 1280*800 and 2048*2048, *ultra high* for superior values

ROM
Read Only Memory. Built-in memory for reading vital information for the working of the processor.

Rotation
Movement where all the points of an object rotate, in the same reference system, of the same angle.

S

Scale (Scaling)
A command that applies a multiplication factor to some or all dimensions of an object. The scaling factor may be different in different dimensions (x,y,z). If the scaling factor is the same in all dimensions, it is called uniform scaling, otherwise it is known as non-uniform scaling.

Scanner
A device that basically copies a paper image into computer memory. The image is digitalized and converted in to bitmap format and sent to the computer.

Segment
Part of a straight line between two points along it.

Selection trap
A panel superimposed on the cursor when the program requires the selection of a graphic primitive. The graphic primitive is selected if you click on it when it’s inside the selection trap.

Serial Port
An output from the computer, from which the eight bits of a data byte are sent one at a time, over a single wire.

Significant digits
The number of significant digits that the system keeps in the database.

Sixtieths of degrees
Usual unit of measurement where the rotation of the angle measures 360 degrees and where the degree contains 60 minutes, each of which is sub-divided in 60 seconds. For example:75° 28'52".

Slant
Angle of slant of the text in relation to the direction of the writing.

Snap
One of the most used options in the CAD system, it doesn't have a equivalence in the manual drawing. It consists of hooking automatically the cursor onto the nearest point of the type specified by the user; for example the middle point of a segment, the center of a circle, a grid point.

Snap point
A point at which the computer can refer to in the snap activity.

Software
The total programs present in the memory, RAM or mass, of the computer.

Spline
A command which is used to interpolate in the best possible way a group of points on the screen (for interpolation we mean, tracing a continuous curve that passes through all the assigned points). BlueCAD uses, for the Spline command, polynomia cubes.

Status Lines
An area of the screen which carries important general information about the active drawing, active layer, color, current co-ordinates of the cursor etc.

Step, hatching
Distance between two hatching lines.

Stretch
A command that allows a primitive to be modified by moving/stretching a vertex in the ways specified by the user.

System
SEE: Platform

T

Transformation
Modification of a graphic primitive by a movement, rotation, scaling or mirroring.

Translation
Moving in space without rotating.

V

Vectorial format
Format used by all CAD systems for which geometric information is essential. If you have to memorize a circle primitive, it is useless codifying the position of the pixel that belong to it: such information wouldn't make sense, if not extensively elaborated, once the view is changed. Insteadin a vectorial format the program memorizes the neccessary geometric information of the circle (for instance center and radius).

Video Card (Video adapter)
Electronic card that controls the video activity.

W

Window Box
SEE: Dialog Box

Window
In an operating system it is a rectangular sub-region of a frame or screen.

Windows 95
Microsoft, mutithreading and multitasking 32 bit operating system for PC.

Word
Identifies a group of 16 bits. Double word is a group of 32 bits.

Workstation
Term that indicates a computer more powerful than a PC, usually it is a 32 or 64 bit with UNIX operating system.

Z

Zoom
Exactly the same as the zoom found on a camera, it establishes the distance that the user wants to view the drawing. It varies the portion of the drawing visible on screen and the degree of detail

Analytic Index

—D—

—E—

—F—

—G—

—H—

—I—

—L—

—M—

—O—

—P—

—R—

—S—

—T—

—U—

—V—

—W—