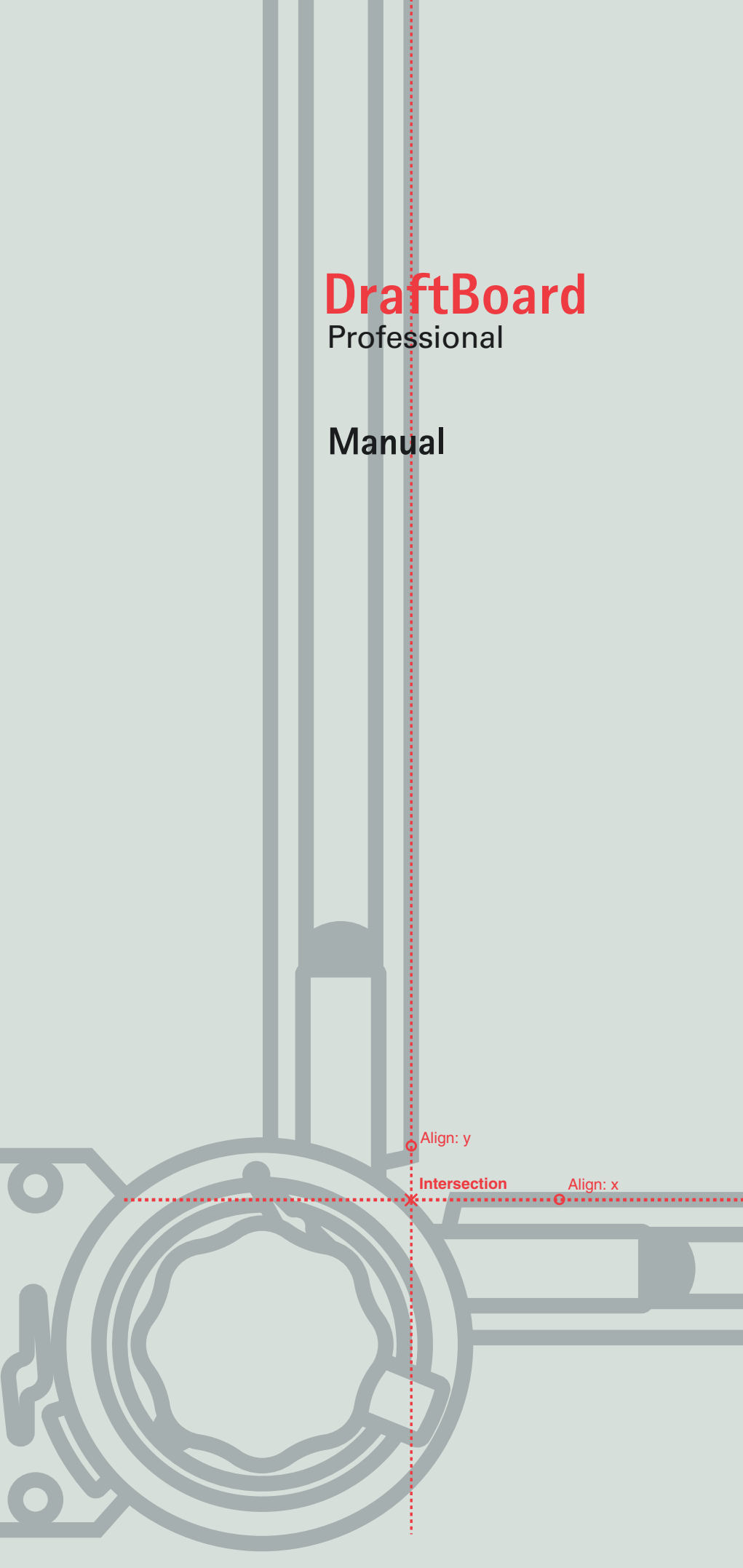
DraftBoard
Professional

Manual



DraftBoard
Professional

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CHAPTER

1

First Steps

Documentation

Registration

System Requirements

Installation

First Steps

This manual describes ***DraftBoard*** *Professional* and ***DraftBoard*** *Professional*.

The *Getting Started* chapter briefly describes the documentation of ***DraftBoard***, tells you how to install the program on your computer and use it with *Microsoft Windows*. It lists the type of equipment you need, tells you how to register your copy of ***DraftBoard***.

Documentation

The following manual describes ***DraftBoard*** for *Windows 2000, Windows XP*, *Windows Vista* and *Windows 7*.

Before using this manual, however, you will need to install ***DraftBoard***. The installation is described in the *Getting Started* chapter. After installation, we encourage you to continue with the *Quick Introduction* chapter and the exercises included in the Tutorial.

Manual Layout

The ***DraftBoard*** documentation consists of the following parts:

- *Getting Started*
- *Reference Guide*
- *Tutorials*
- *Appendices*

Tips and advanced user notes are displayed in the margin throughout the manual. You don't need to know this information to run ***DraftBoard***.

Introduction

The *Introduction* part exist of the chapter *First Steps* with informations about the Documentation and the Installation of ***DraftBoard*** and the chapter *Quick Introduction*.
The *Quick Introduction* chapter gives you general information about the program. It describes the basic components of ***DraftBoard***—the mouse, window features, menus and dialog boxes. It gives a brief overview of useful features and may be all you need to know if you are familiar with CAD software.

Reference Guide

This manual describes how to use ***DraftBoard*** including all its pertinent tools and commands. If you want information on a particular tool or command, simply look it up in the *Index*.

Detailed informations about the user interface of your computer you find in the *User Guide* for *Microsoft Windows*.

Tutorials

The Tutorial provide step-by-step illustrated instructions to help you to become familiar with ***DraftBoard's*** features and functions. It shows you how to use common drafting and design practices on a computer. You can learn about ***DraftBoard*** by doing the exercises in the Tutorial. If you'd rather, you can read the *Reference Guide* first and then go through the *Tutorial* to put what you've learned into practice.

Appendices

At the end of this manual you find several appendices with additional information about specific settings for some ***DraftBoard*** *components*.

Operators and Units

This section describes all *Operators* and *Units* which are accepted by all ***DraftBoard*** entry boxes.

Special Characters

This section lists all the special characters and symbols you can use in ***DraftBoard*** which are not directly available from the keyboard.

Date Interfaces

This appendix contains additional information about the *Data Interfaces* integrated into ***DraftBoard***.

Plotter Fonts

This section lists all plotter fonts available in ***DraftBoard*** that you can use in addition to the *PostScript* and *TrueType* fonts that are installed on your computer.

Hatch patterns

DraftBoard contains a library with many ***ISO*** and ***DIN*** *hatch patterns*. This appendix lists all with ***DraftBoard*** available hatch patterns and describes as how you can define your own hatch patterns and add them as ***.pat*** *files* to the ***DraftBoard*** *hatch library*.

Registration

You find your valid **Serial Number** on your Registration card or on the Software box.

Contrary to popular opinion, registration cards and trash aren't synonymous. In fact, the registration card for ***DraftBoard***, conveniently located inside your software box, is quite valuable – to you and to us. We'd really like to encourage you to return it. Filling out the registration card is a painless process.

For such a minor investment of time, you'll become a registered customer. Only by registering ***DraftBoard***, are you entitled to receive Telephone product support and Free updates

System Requirements

You need the following equipment to run ***DraftBoard***:

- Pentium based processor or better
- Arbeitsspeicher*:* **256 MB** (512 r better recommended)
- a hard disk with **30 MB** of unused space
- a **CD ROM** drive for installing the software
- *MS Windows* compatible Graphic card
- **Windows:** *Windows 2000*, *Windows XP, Windows Vista* or *Windows 7*

Installing DraftBoard

DraftBoard for *Windows* comes on a CD-ROM.

Installing a Single User License for Windows

1. Start *Microsoft Windows*.
2. Insert the ***DraftBoard*** *CD ROM* into the CD ROM drive of your computer.
3. Start the *Explorer* .
4. In the *Explorer* click on the icon for your CD ROM drive.
   The content of the CD ROM is displayed.
5. Start the installation by double clicking the file **Setup32.exe**.
6. Follow the directions on the screen.
7. During the installation you are asked to enter your ***Authorization*** *Code*, that you will find on the enclosed *License Agreement*.
8. When the installation is complete, click the **OK** button.
   The installation automatically creates the submenu **DraftBoard** in the **Program** menu of the *Windows* **Start** menu.
   This menu contains besides other entries the menu item

   DraftBoard Professional 4.10

   that will start ***DraftBoard***.

You should read the **Read me** file first that you can open with the **Readme** menu item, since it contains information that was not available when this manual was printed.

CHAPTER

2

Quick Introduction

Using a Mouse

*Parts of the **DraftBoard** Window*

Preferences

Quick Introduction

This chapter describes the basic components of ***DraftBoard***—the mouse, window features, menus, and dialog boxes. This is a brief overview of useful features and may be all you need to know if you are familiar with CAD Software.

If you need more information about standard Windows elements such as menus, scroll bars, File menu commands, and dialog boxes, refer to your *Microsoft Windows User's Guide*.

Using a Mouse

The mouse is your communication device; you use it to tell the computer what you want to do. You will use the mouse to indicate locations, choose commands, select tools, and construct objects.

If your mouse has more than one button, you will use only the left button with ***DraftBoard***.

If you are left-handed and if your mouse has more than one button, you may want to change the functionality to the right mouse button. You can make this change in the **Control Panel** of *Windows*.

This manual uses the following terms for mouse activities:

Pointer An arrow or other graphic symbol that allows selection or creation of an object. You move the pointer to point to a command or an object on the screen. Depending on its location, the pointer is an arrow or looks like the current tool.

Arrow Pointer

DraftBoard Selection Arrow

Center-Point *Circle* Arrow

To move the pointer, move the mouse on the mouse pad. You use several different mouse actions with ***DraftBoard***.

Point Move the mouse until the pointer is over the item you want.

Press Press and hold down the mouse button.

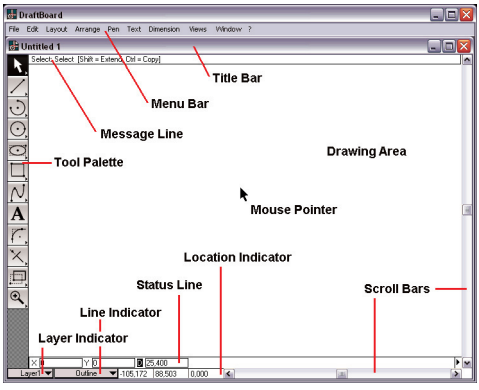
Click Quickly press and release the mouse button once.

Double-click Click the mouse button twice, quickly in succession.

Drag Press and hold down the mouse button, move the mouse, then release the mouse button.

Parts of the DraftBoard Window

When you start ***DraftBoard***, the following window appears:



Title Bar	Includes the title of the active document and two buttons: the *Zoom box* and the *Close box*, for controlling the window.
Menu Bar	Contains the ***DraftBoard*** menus of commands and settings. You can make choices from the menus with the mouse or by using special key combinations.
Message Line	Displays the name of the current tool and step-by-step instructions for using the tool.
Tool Palette	Contains the drawing and editing tool icons you will use for constructing, editing, and annotating geometry.
Drawing Area	Consists of multiple layers where you construct and annotate geometry.
Pointer	Shows the active position on the screen. If the pointer is in the drawing area, its shape represents the current tool.
Layer Indicator	Displays the name of the current layer and provides a menu for changing the work layer.
Line Indicator	Displays the name of the line style and provides a menu for changing the line style.
Location Indicator	Shows the x,y coordinates of the pointer location.
Status Line	Shows the coordinate location and other geometric parameters of the current construction.
Scroll Bars	Allow you to move around a drawing so you can see different sections of it through the ***DraftBoard*** *window*. The scroll buttons allow you to move one line at a time.

Tool Palette

The tool palette is a group of tool icons along the left side of the screen. The icons represent the tools for drawing, editing, and annotating geometry.

Selecting a tool from the tool palette

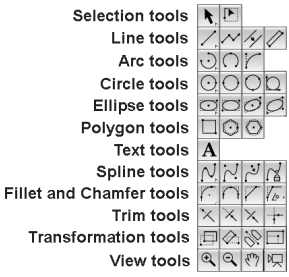
1. Position the arrow pointer on the icon of the tool you want to use.

2. Click the mouse button.

The icon appears highlighted to indicate its selection. The **Single Line** tool is selected above.

Tool Subpalettes

Most of the tools in the tool palette contain a subpalette of tools with related functions. The ▶ in the lower-right corner of the tool icon shows the presence of a subpalette which contains related tools.

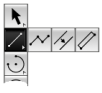


Viewing and selecting from a subpalette are similar to choosing a command from a menu.

Selecting a tool from a subpalette

1. Position the arrow pointer on the tool.
2. Press and hold down the mouse button.

   The subpalette appears to the right of the tool.



3. With the mouse button pressed, ***drag*** the pointer to highlight the desired tool.

4. Release the mouse button.

   The selected tool replaces the previous tool in the tool palette. The highlighted icon in the tool palette shows that your selection from the subpalette is the active or current tool.

   If you want to select a tool that is already displayed in the tool palette, you only need to click it; you don't need to select it from the submenu.

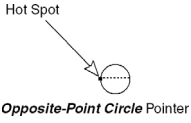
   The new tool is visible in the tool palette until you select another tool from the same subpalette. The tools on the subpalette remain in the same order; only the tool displayed on the tool palette changes.

Once you select a tool, additional information appears to help with your construction. The pointer, pointer locator, message line, and status line all provide feedback about the active tool.

Smart Pointer

When you select a tool and move the pointer into the drawing area, the pointer shape is representative of the tool.

Some of the pointers, like the single line pointer, are simple crosshairs. Others, such as the ***opposite-point circle*** pointer, resemble the tool itself.

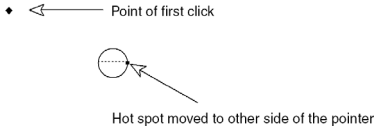


Opposite-Point Circle Pointer

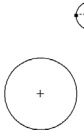
The pointer, called a *smart pointer* displays indicators for multi-step procedures. Each smart pointer has a dot, the *hot spot*, showing the

next point you should specify. The dot changes position on the pointer during each step of the construction.

For example, the ***opposite-point circle*** pointer illustrated above shows that the first click of the mouse places a point on one edge of the circle you're creating. After you click a location, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite edge of the circle.



After you click the second location, the circle appears. The hot spot moves back to its original position on the pointer so that you can create another circle.



Location Indicator

The number of decimal places displayed in the indicator box is determined by the **Precision** setting in the **Units** dialog box (under **Preferences** in the **Layout** menu).

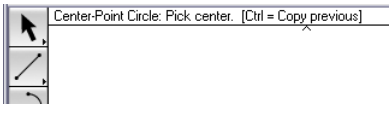
The location indicator is two numbers to the left of the horizontal scroll buttons at the bottom of the drawing area.

This indicator continuously tracks the pointer location when the pointer is in the drawing area, displaying the X,Y coordinates of the current location relative to the origin. The origin (**0,0**) is in the center of the screen when you open a new document. A symbol appears at the origin (0,0) when you make the grid visible by choosing **Show Grid** from the **Layout** menu.

Message Line

The message line across the top of the drawing area provides concise instructions for the use of the current tool.

For example, after selecting the ***Center-Point Circle*** tool, the message line appears as illustrated below:



The instructions in the message line for some tools also indicate optional activities. For example, if you hold down the **Ctrl** key while using the ***Center-Point Circle*** tool, the next mouse click creates a copy of the last circle with the center placed where you clicked.

Status Line

The status line provides measurements, angles, X,Y coordinates and delta values for the current construction. The current tool determines the number of status boxes and which of the status boxes is highlighted after the construction. For example, if you select the ***Center-Point Circle*** tool, the status line shows the X,Y coordinates for the center of the circle and the length of the diameter.

The number of decimal places displayed in the *Status Boxes* is determined by the **Precision** setting in the **Units** dialog box (under **Preferences** in the **Layout** menu).

You can enter up to *10 characters* in each *Entry Box* of the *Status Line.*

When you click the last point of the circle, the diameter (**D**) entry box highlights in the status line to indicate that it is active. It shows the diameter of the circle you just created. If you type a new number, and press the **Enter** key, the diameter of the circle you just created changes.

You can change any or all entries in the status line, but when you press the **Enter** key, you can't make any more changes in the status line.

***Important**: Pressing the Enter key again would create an identical object at the same location. Later you can only modify objects using the Edit Objects command in the Edit menu.*

Moving between status boxes

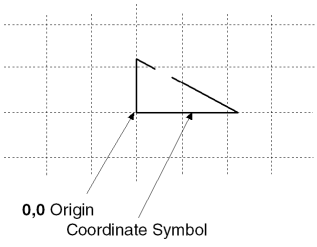
You can use the **Tab** key to move to the right, highlighting the next box. When you press the **Enter** key, the construction redraws according to the new specifications in the status line.

Drawing Area

You use the drawing area for all construction, editing, and annotation of geometry. You can think of the drawing area as a sheet of paper of unlimited size that you can use to construct full-size unscaled drawings. You can use the scroll bars to move the sheet so the portion you want to work on is visible in the window.

Displaying the grid

If you wish to work with a grid in the drawing area, choose **Show Grid** from the **Layout** menu.



When the grid is visible, constructions *snap to the grid*, meaning that any geometry point that you click snaps onto the closest grid point. The coordinate symbol appears at the origin when the grid is visible.

Normally, you will not need to display the *Grid*, since the *Drawing Assistant* offers a more elegant support than the grid.

Scroll Bars

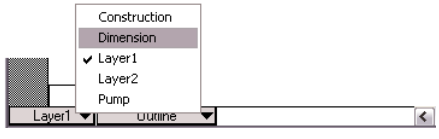
The scroll bars allow you to move the sheet up and down or right and left. You can display different parts of the drawing sheet by ***dragging*** the slider of a scroll bar to the approximate location. For example, the right, center, or left position in the horizontal scroll bar displays the right side, middle, or left side of the drawing.

You can also click the arrows at the end of the scroll bars to move the sheet one line at a time. If you click in the scroll bar, the sheet moves one window at a time.

Work Layer Indicator

The *Work Layer* in the lower-left corner of the screen shows the current layer. New geometry goes on the *work layer*. If you want your construction to go on a different layer you must change the current *work layer*.

To select a new *work layer*, position the pointer over the Layer indicator, then press and hold down the mouse button. All available layers will be displayed in a pop-up menu.

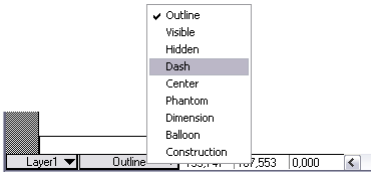


You move the mouse pointer with the left mouse button still pressed over the layer which should become the work layer and release the mouse button. The selected layer becomes the new work layer.

Pen Indicator

The *Pen Indicator* in the lower-left corner of the screen shows the current pen. If you want to use a different line style you must change the current *pen*.

To select a new *pen*, position the pointer over the pen indicator, then press and hold down the mouse button. All available pens will be displayed in a pop-up menu.



You move the mouse pointer with the left mouse button still pressed over the pen which should become the current pen and release the mouse button. The selected pen becomes the new line style.

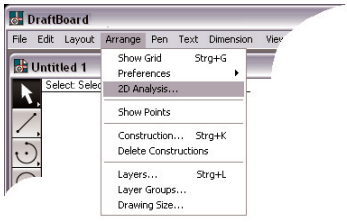
Menu Bar

The ***DraftBoard*** menus contain related commands and settings.

File	Commands that affect entire documents (files).
Edit	Commands to select and manipulate objects.
Layout	Commands and settings that specify the drawing area and provide program features and functionality, such as construction lines and 2D analysis.
Arrange	Commands for zooming to change the area displayed in the window. You can also set specifications for objects.
Pen	Commands to specify pen characteristics: *style*, *color*, *weight*, and *pattern*. *Crosshatching* and *arrows* are also available on this menu.
Text	Commands to set the *font*, *size*, *style*, *alignment*, and *indentation* of text.
Dimension	Commands that specify dimensions and their format and tolerance.
Windows	Lists all drawing currently open in ***DraftBoard.*** The drawing in the foreground is marked by a checkmark.
?	The **Help** menu provides access to the *Reference Guide* and *Tutorial* of ***DraftBoard***.

Displaying a menu

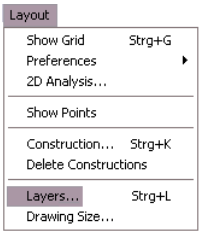
1. Position the **arrow pointer** on the **menu name**.
2. Click the mouse button.



The menu appears. If you want to dismiss the menu without making a choice, click outside the menu.

Choosing a command from a menu

1. Position the **arrow pointer** over the **menu name**.
2. Press and hold down the mouse button.



3. With the mouse button still pressed, ***drag*** the pointer down the menu. Each command highlights as the pointer moves over it.
4. When the command you want highlights, release the mouse button.

   The command executes, or the setting, such as **Selectable Points**, toggles *on* or *off*.

Mouse versus Keyboard

DraftBoard's menu items can be chosen with the mouse or with a combination of keys on the keyboard.

For example, you can use with *Windows* any of three methods for displaying the **Edit** menu:

- Click on **Edit** in the menu bar.
- Press the **Alt** key and then type **E** (for **E**dit).
- Press the **Alt** key and then press the right **Arrow** key until **Edit** is highlighted in the menu bar; then press **Enter**.

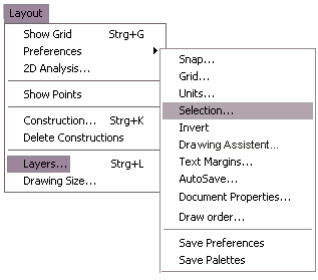
You have three methods for choosing commands with the keyboard. For example, you could use any of the following methods to choose the **Show Grid** command from the **Layout** menu:

- Press **Alt** and **L** (for **L**ayout) and then type **G** (for Show **G**rid).
- Press **Alt** then use the right **Arrow** key to highlight **Layout** and press **Enter**.
- Hold down the **Ctrl** key and type **G** (for Show **G**rid).

The first method is the *mnemonic method*. You press the **Alt** key with the appropriate letters for the menu and command as indicated by the **underlined** character in the names. The third method is a *keyboard accelerator* and, when available, is denoted by the *key sequence* listed on the menu.

Submenus

Commands followed by a ▸ symbol have submenus which display when the command is highlighted.



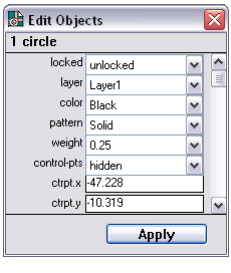
Just as good is to click on a menu, press the mouse button and ***drag*** the pointer to a menu item containing a submenu. In the submenu you ***drag*** the mouse pointer to a command. As soon as you release the mouse button the selected command is executed.

1. Pull down the menu.
2. ***Drag*** to highlight a command followed by a ▸ symbol.

   The submenu displays.
3. With the mouse button still pressed, ***drag*** over to the submenu.
4. ***Drag*** down the submenu until the desired command is highlighted.
5. Release the mouse button.

Dialog Boxes

When you choose a command followed by an ellipsis ▶, such as **Edit Objects** on the **Edit** menu, a dialog box appears.



Win

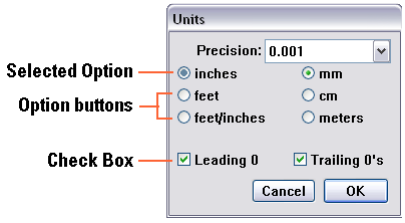
Some dialog boxes can be closed by clicking on the **Close box** in the *Title bar*.

Dialog boxes allow you to qualify the command you chose by adding information. For example, in the **Edit Objects** dialog box above, you can change the specifications of the selected object.

If a dialog box obscures your view of the drawing area, you can ***drag*** it with the pointer on the **Title bar** to a new location.

Option Buttons

Option buttons indicate mutually exclusive choices; you can select only one option at a time (as you might select a station on your car radio). Click the option you want and the button turns black, as shown by the inches option above.



Check boxes

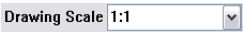
Check boxes provide options you can switch on and off and which are not mutually exclusive. An X shows the option that is set.

List/Entry boxes

Some dialog boxes contain lists of options, displaying a drop shadow to show that a pop-up menu exists. Move the pointer to the entry box and press the mouse button to display the list.

The item you highlight appears in the box once you select it.

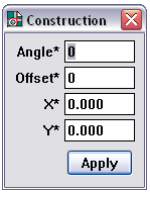
Some list boxes as the **Scale box** in the **Drawing Size** dialog box allows you to type an entry as well as choose from the pop-up menu.



To type an entry, select the current entry (if it isn't already selected), then type a new entry. The new entry replaces the original entry.

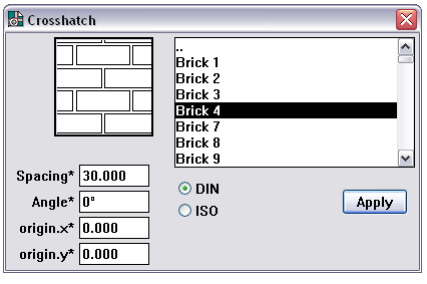
Asterisks

When an item in the dialog box displays an **asterisk** (*****), you can specify a value by clicking or ***dragging*** in the drawing area. This feature is particularly useful for specifying location because you don't need to know any x,y coordinates.



Apply Buttons

Some dialog boxes have an **Apply** button that allows you to apply the specification you just set. You can leave the dialog box open to set other specifications.



For example, once you crosshatch a part, you could leave the **Crosshatch** dialog box open and select other objects to be crosshatched.

Closing a dialog box

If a dialog box contains an **OK** or **Cancel** button or an action button such as **Open**, the dialog box closes when you click the button. Otherwise, you dismiss the dialog box by clicking the **Close box** in the upper-left corner of the box. If the dialog box has an **Apply** button, such as the **Crosshatch** box, you must click the **Close** box to close it.

Toggling Commands

Commands that set a condition (such as **Selectable Points** and **Arrow At Start**) display a checkmark ✓ in the menu to indicate that they are active. To turn a command off, choose it and the checkmark will disappear.

In the case of pen styles and text characteristics, the check shows the current setting.

Other commands, **Show Grid**, **Show Points**, and **Show Palette**, toggle to **Hide** (*Grid*, *Points*, or *Palette*) when the component is visible.

Preferences

All files are saved with the settings as they are established when you close the file. The default characteristics used for new files are contained in the preferences file. The preferences file name is *Prefs.vlm*. This file must be in the **DraftBoard** folder.

If you want to use different settings for your work, you can change and save the default settings using the **Save Preferences** command in the **Layout** menu. Every new document will then open with the new default settings. The following specifications can be set in the preferences file:

- Pen styles
- Text characteristics and margins
- Preferences settings (*snap*, *grid*, *units*, and *selection indicator etc.*)
- Grid display
- Layer and sheet specifications
- Work Layer
- Dimension and tolerance formats
- Arrowhead type and display
- Drawing size and scale
- Zoom scale
- Fillet radius
- Chamfer angle and length
- User defined colors
- Document Properties
- AutoSave settings

Changing the default settings

1. Create a drawing with the settings you want for each new drawing.
2. Choose the **Save Preferences** command in the **Preferences** submenu of the **Layout** menu.

   The preferences are set for all subsequent new documents.

CHAPTER

3

Creating Geometry

Drawing Techniques

Drafting Aids

Drawing Tools

Creating Geometry

This chapter describes how to create the geometric objects that are the building blocks for your designs. It also describes how the *Drawing Assistant* helps you design faster and easier. The following topics are contained in this chapter:

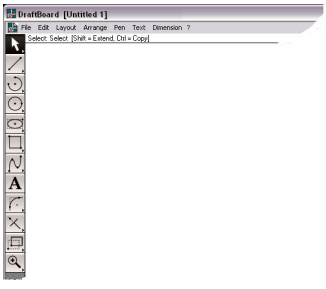
- Drawing Techniques
- Drafting Aids
- Drawing Tools

This chapter requires that you know how to *open* or *save* a file or how to use a *Zoom-tool*. If you are unfamiliar with *Windows* programs see the *Getting Started* section in the *Tutorial*.

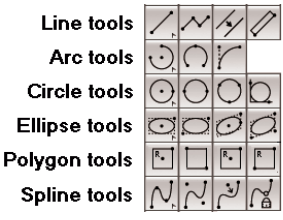
You will find more about the functionality of the ***Drawing Assistant*** in *Chapter **3***.

Function Palette

Drawing begins with the Tool palette, located at the left of the ***Draft-Board*** drawing area shown below:



The Tool palette has six subpalettes of tools that you can use to create geometric objects. These six tools are the drawing tools.



Drawing Techniques

Most drawing tools allow two methods of creating an object: clicking and ***dragging***.

Clicking This method allows you to create objects by clicking points with the mouse. The benefit of this technique is that you can perform unrelated activities, such as zooming, while you are creating the geometry.

Dragging The ***drag*** method allows you to indicate two points by ***dragging*** from one to the other. Press and hold down the mouse button to set the first point. Next, ***drag*** the mouse to the location for the next point and release the mouse button. The benefit of this technique is that a *rubberband* image appears during the construction to show you how the object will appear once you release the mouse button.

You can use either or both methods to construct most objects.

To help make object creation easier, ***DraftBoard's*** tools contain a feature we call *Smart Pointers*. As you work with a tool, the tool icons show you the points you must indicate to create an object with the tool.

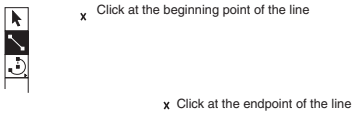
Each dot on the icon represents a point you must place either by clicking or by ***dragging***. In addition, the *Smart Pointer* indicates the order for designating points as you are drawing the geometry.

Creating a line by clicking

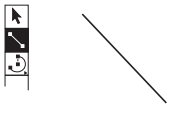
1. Select the **Single Line** tool from the Tool palette.

2. Move the pointer to the drawing area and click to set the starting point of the line.
3. Move the pointer to the desired location for the endpoint of the line and click.

If you use the clicking method to draw a line no *rubberband* line guides you.



The line is drawn between the two points.



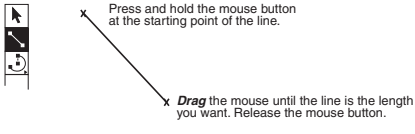
Immediately after construction, you can make changes in the Status Line to alter the length or position of the line. If the geometry isn't satisfactory, just press the **Backspace** key.

Creating a line by *dragging*

1. Click the **Single Line** tool in the Tool palette.

2. Move the pointer into the drawing area.
3. Press and hold down the mouse button at the desired location for the starting point of the line.
4. With the mouse button still held down, move the mouse to the desired location for the endpoint of the line.

Notice that when you ***drag*** between points, a *rubberband* line guides you. If you use the clicking method, you can perform unrelated activities, such as zooming, between clicks.



5. Release the mouse button to set the endpoint.
   Immediately after construction, you can make changes in the *Status Line* to alter the length or position of the line. If the geometry is not satisfactory, just press the **Backspace** key.

Drafting Aids

For creating geometry objects the ***DraftBoard*** user interface provides the following drafting aids:

- Drawing area
- Message Line
- Smart Mouse Pointer
- Status Line
- Drawing Tools

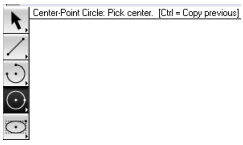
Drawing Area

All objects are created, modified and annotated on the drawing area. You can imagine this drawing area as an infinite large sheet of paper on which you can create any type of geometry object at its original size. Using the Scroll bars on the left and lower side of the drawing window you can move the sheet up and down or right and left. You can display different parts of the drawing sheet by ***dragging*** the slider of a scroll bar to the approximate location.

Message Line

The Message Line across the top of the drawing area provides concise instructions for the use of the current tool.

For example, after selecting the **Center-Point Circle** tool, the Message Line appears as illustrated below:



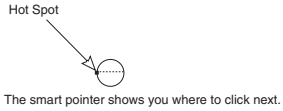
The instructions in the Message Line for some tools also indicate optional activities. For example, if you hold down the **Control** (**Ctrl**) key while using the **Center-Point Circle** tool, the next mouse click creates a copy of the last circle with the center placed where you clicked.

Smart Pointer

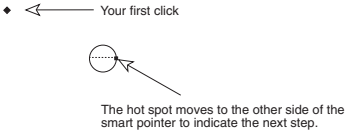
When you select a tool and move the pointer into the drawing area, the pointer shape is representative of the tool.

Some of the pointers, like the single line pointer, are simple cross-hairs. Others, such as the **Opposite-Point Circle** pointer, resemble the tool itself.

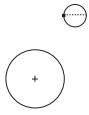
The pointer, called a *smart pointer,* displays indicators for multi-step procedures. Each smart pointer has a dot, the *hot spot*, showing the next point you should specify. The dot changes position on the pointer during each step of the construction.



For example, the **Opposite-Point Circle** pointer illustrated above shows that the first click of the mouse places a point on one edge of the circle you're creating. After you click a location, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite edge of the circle.



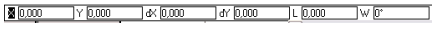
After you click the second location, the circle appears. The hot spot moves back to its original position on the pointer so that you can create another circle.



The Status Line

Whenever you select a tool from the Tool palette, the *Status Line* appears along the bottom of the drawing area. It contains boxes giving information about the current construction. For example, when the **Single** or **Connected Line** tool is the current tool, the *Status Line* contains boxes for the *x and y coordinates* of the beginning point and the *x* and *y values* for the endpoint of the line. It also contains the value of the *length* and *angle* of the line.

You can set the **number of decimal places** or fractional round off with the precision entry in the **Units** dialog box from **Preferences** in the **Layout** menu.



You can use the Status Line in three ways:

- To create an object with keyboard entries only.
- To edit an object that was just created and is still selected.
- To create an additional object using the current tool.

Whenever you construct an object, the status box containing the specification that you are most likely to change is selected (the box is highlighted).

For example, when you draw a line with the **Single Line** tool, the **Length** box is highlighted so you can enter a value for the length. Just type a value. When you press **Enter** the line is redrawn at the new length.

Using the Status Line

You can enter up to 10 characters in each **Entry field** of the Status line. Notice how each method affects the selection of the contents of the status box. Clicking once inside the box places the cursor where you clicked. If you double-click inside the box, the entire entry is selected so you can change it.

If you want to make an entry in a different status box, you can use one of four selection methods:

- Press the **Tab** key to cycle the selection highlight through the status boxes from left to right.
- Click inside the status box and the pointer becomes an **I-beam** text cursor.
- Click the box label to select the entire contents of the box.
- Double-click inside the status box to select the entire contents of the box.

Using the Status Line with Drawing Tools

Once you have drawn an object, you can adjust its specifications such as *length*, *angle*, and *location* in the status Line. You must make these changes immediately, before you construct another object, select a different tool, or choose a command. You can make only one series of changes in the status line (in as many boxes as necessary); after you press the **Enter** key ***DraftBoard*** redraws the object to your specifications. Once you have pressed the **Enter** key, you must make subsequent changes with the **Edit Objects** command in the **Edit** menu.

Try the following exercises to create and change a single line with the Status Line.

Altering geometry in progress with status boxes

1. Select the **Single Line** tool.
2. Click **2** locations in the drawing area.
   The length (**L**) box is highlighted in the status line.
3. Type **3**.
   The *3* is entered directly in the **L** box.
4. Press the **Tab** key to select the next status box.
   The angle (**A**) box is now highlighted.
5. Enter **15**.
6. Press the **Enter** key.
   Pressing the **Enter** key completes the data entry for this object. The line is redrawn **3** units long and at a **15°** angle.

Creating additional geometry with status boxes

1. With the **Single Line** tool still selected from the previous example, click **2** more points.
2. Type **4**.
3. Press the **Tab** key and type **25**.
4. Press **Enter**.

   Another line is drawn.

You must remember that when you press the **Enter** key, ***DraftBoard*** constructs the object based on the specifications in the status boxes. Pressing **Enter** a second time in the above example would create a second line with the same values. Since the lines have the same values, the second line overlays the first line and you can't see it.

Creating new geometry with status boxes

1. Select the **Selection** tool so the **Single Line** status box clears.
2. Click the **Single Line** tool again.

   The **X** status box is active, showing that you can enter a value for the X coordinate of the beginning point of the line.
3. Type **0**.

   Note: Do not press **Enter** until you reach step **#10**.
4. Press the **Tab** key.

   The **Y** box is highlighted.
5. Type **0**.
6. Press the **Tab** key.

   The **dX** box is highlighted. The dX value is the delta X, the numeric difference between the beginning and ending X coordinate.
7. Type **2**.
8. Press the **Tab** key.

   The **dY** box is highlighted.
9. Type **2**.
10. Press the **Enter** key.

    The line is drawn.

The *angle* and the *length* of the line are calculated by the values you have specified. If you specify values for the *angle* and the *length* of the line you don't have to enter values in the **dx** and **dy** status fields and vice versa.

Creating Geometry Offset from a Point

If you want to create some geometry that is offset from an existing point, you can use the *Status Line* to specify the offset.

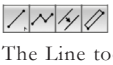
Creating geometry offset from a point

1. Select the tool you want to use.
2. Move the pointer over the *control point* from which you want the offset.
3. Click once to lock onto that point.
4. Click in the appropriate **X** or **Y** box in the Status Line, placing the text cursor at the end of the entry.
5. Type the offset (such as **+ 3"**) and press **Enter**.
6. Continue with your construction.

Drawing Tools

The following section describes the drawing tools found on the *Tool palette*.

Line Tools



The Line tools on the Tool palette create *line segments*, *connected lines*, *lines parallel* to existing lines, and *smart walls*. As you create a line, the coordinate locations, line length, and angle from horizontal appear in the Status Line.

The line is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Single Line Tool

This tool draws a line between two points. You can click or ***drag*** to draw a line.

Using the Single Line tool

- Click two endpoints of the line.

  Or:

- ***Drag*** to indicate the endpoints of the line; press at the beginning and release at the end of the line. As you ***drag***, you see a *rubber-band* line that previews your construction.

Notice that after completing a line the Message Line says **Ctrl = Copy**. This indicates that if you hold down the **Ctrl**-key and click once in the drawing area, a line will appear that is identical to the one just drawn. The location of the click designates the location of the first point.

The Status Line allows you to specify the **X, Y** *coordinates* of the beginning, the relative location of the end (*delta X* and *delta Y*), the line *length*, and the *angle* from horizontal. Once a line is drawn, the line **Length** is the selected status box.

Drawing a line perpendicular to another object

1. Construct the line (or spline).
2. Move the pointer to the object until a Drawing Assistant ***on*** notation appears.
3. ***Drag*** straight away from the object in a ***perpendicular*** direction.
4. ***Drag*** the extent of the line.

Drawing a line tangent to or perpendicular to a curve

1. Construct an arc, circle, or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drawing Assistant ***on*** notation appears. The Drawing Assistant notation must be ***on*** rather than ***endpoint***, ***quadrant***, or ***midpoint***.
4. ***Drag*** in the appropriate direction (straight out for perpendicular or at an angle for tangent) until the Drawing Assistant ***perpendicular*** or ***tangent*** notation appears.
5. When the Drawing Assistant locks on to ***perpendicular*** or ***tangent***, you can then ***drag*** the line around the arc to the location you want and extend the line to the length you want.

Creating a point

You can create a point by creating a line and setting its length to zero. Set the starting point of the line with the mouse, then type **0** in the **Length** entry field of the status line and press the **Enter** key. The line ("point") displays a **+** to indicate its location.

You can also double click the **Single Line**, the **Center-Point Circle** or the **Opposite-Point Circle** tool in the same spot to create a point. These points can be selected and deleted as any other point. These points that are displayed by a **+** on the screen are printed as points.

When creating a point with the **Single line tool** by clicking twice the same position you have to select a different tool before you continue to draw. Otherwise you would create a line with a double *Starting point*. This is not required when you use the **Circle tool** to create a point.

Connected Lines Tool

This tool draws lines in which the endpoint of one line segment is the beginning point of the next.

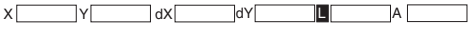
Using the Connected Lines tool

1. Click or ***drag*** to indicate the endpoints of the line segments.

   If you click a point and then change your mind, press the **Escape** key to remove the last line. Pressing the **Delete** or **Backspace** key or choosing **Undo** removes all connected lines in the current construction.
2. Indicate the last point by double-clicking or by choosing another tool.

If you want to create a connecting arc between two points rather than a line, hold down the **Ctrl** key *and* then click the endpoint of the segment. You must have constructed one line segment before you can use the arc function.

The radius of a arc created with the **Connected line tool** can be modified with **Edit objects** command.



The status line allows you to specify the *X*, *Y* and *Z coordinates* of the beginning, the relative location of the end (delta X and delta Y), the line *length* and the *angle* from horizontal. Once the line segment is created, line **Length** is the selected status box.

Parallel Lines Tool

This tool constructs lines parallel to existing lines. Use one of the following methods for creating parallel lines:

Using the Parallel Lines tool

- Select the **Parallel Lines** tool, ***drag*** the line; press to select the line, and release to indicate the position for the parallel line. A rubberband line moves with the pointer.

  Or

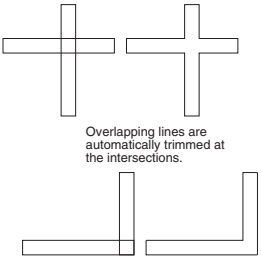
- Select the **Parallel Lines** tool, click the line you want to duplicate, and then enter a delta distance in the status line and press **Enter**. (This method does not allow you to control on which side of the original line the new line is drawn.)

The status box allows you to specify the **distance** between the selected line and the parallel line.

The second method allows you to control the direction of the offset by entering **+** or **-** in front of the indicated values. Entering **+** offsets the object to the *right* or *downwards* on the screen, entering **-** to the *left* or *upwards*.

Smart Wall Tool

This tool works in much the same way as the **Single Line** tool except that it draws double lines or **walls**. Walls created on the same layer are automatically trimmed to where they touch or intersect.



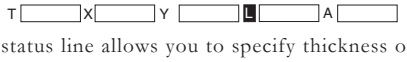
Automatic trimming of smart walls occurs only when the walls are on the same layer.

As you ***drag*** the mouse, the defining side of the wall falls on the construction line you drag along. The other side of the wall automatically fills in at the thickness you specify in the status line. The position of the automatic wall depends on your last activity. If the second wall is on the wrong side of the construction line, press the **Shift** key to flip it.

Using the Smart Wall tool

1. Select the **Smart Wall** tool.
2. In the status line, specify the thickness (**T**) of the wall.
3. Click or ***drag*** to indicate the ends of the wall segment.
   You can press the **Shift** key to flip the wall to the opposite side.
4. Continue specifying both endpoints for wall segments, as needed.

If you delete a merged segment, the remaining walls redraw.



The status line allows you to specify thickness of the wall, the **X**, **Y**, **Z** coordinates of the beginning point, the wall length and the angle from horizontal. Once the wall segment is created, wall **Length** is the selected status box.

If you want to *fillet* or *chamfer* smart walls you have to ungroup them with the **Ungroup** command in the **Arrange** menu.
It is recommended that you place all symbols, before you *ungroup* and *fillet*/*chamfer* the walls, since ungrouped walls loose their smart features such as automatic trimming. Select only the wall segments not any placed symbols if you want to ungroup a wall.

Wall symbols

Parametrized symbols with an underlying wall segment like the door symbols you find in the *Architect Library* will automatically cut wall segments where they are placed. When you move the symbol along the wall to another position the wall will automatically be closed at the old position and be cut open at the new position.

If you want to use the **Smart Wall** feature for single lines, you have to create the lines with the **Smart Wall** tool and enter **0** for the thickness of the wall.

Arc Tools

You can use the Arc tools on the tool palette to create an arc by any of three methods:

- *Center-Point Arc*
- *3-Point Arc*
- *Tangent Point Arc*

As you create each arc, the status line displays entries such as coordinate locations, radius, angle from horizontal, and delta angle.

The arc is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Center-Point Arc Tool

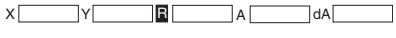
This tool draws an arc from three points: the center point, arc beginning point, and arc endpoint.

Using the Center-Point Arc tool

1. Click the ***center*** point of the arc.
2. Click the radius and beginning of the arc.
3. ***Drag*** the extent of the arc, pressing at the beginning point of the arc and releasing at the endpoint.

You can simply click all three points, but the *rubberband* arc doesn't appear when you use this method.

To construct an arc greater than **180°**, you must ***drag***, rather than click, the endpoint.



The status line allows you to specify the *X, Y coordinates* of the center of the arc, the *length* of the radius, the starting *angle* from horizontal, and the delta angle from the start. **Radius** is the selected status box.

3-Point Arc Tool

This tool draws an arc through the points you select.

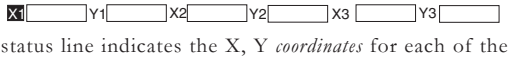
Using the 3-Point Arc tool

1. Click the first position on the arc.
2. ***Drag*** or click the second and third positions. (***Dragging*** displays the *rubberband* arc.)

The arc is drawn from the first position that you indicate in the direction of the next point.

If ***DraftBoard*** can't create the arc tangent to an object it will define an alternate construction.

If you click ***on*** an object while holding down the **Ctrl** key, the arc is drawn tangent to the specified object rather than through the point. You may combine the placement of points to create an arc through a specific point of one object and tangent to another object, or an arc tangent to three objects, or any other combination.



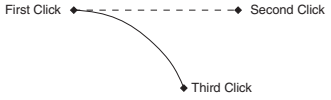
The status line indicates the X, Y *coordinates* for each of the three points.

Tangent-Point Arc Tool

This tool draws an arc beginning at the first point you specify. The second point you specify is the direction vector, and the third point you specify indicates the endpoint of the arc. Essentially, the **Tangent-Point Arc** tool first creates a line, then creates an arc tangent to the line, then erases the line.

Using the Tangent-Point Arc tool

1. Click the starting point.
2. Click the tangent vector. (The arc is tangent to a line drawn from the starting point to this point.)
3. Click the endpoint. The arc is drawn between the first and last point you click and tangent to the line between the first and second points.



The status line allows you to specify the X, Y *coordinates* of the endpoints of the arc and the *angle* of the tangent line.

Circle Tools

The Circle tools on the tool palette construct circles by any of four methods:

- *Center-Point Circle*
- *Opposite-Point Circle*
- *3-Point Circle*
- *Tangent-Point Circle*

Center-Point uses the center and diameter of the circle. *Opposite-Point* uses the diameter. **3-Point** uses three points or can be tangent to existing objects. *Tangent-Point Circle* draws a circle tangent to two objects, using the diameter.

The circle is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

If you want to ***drag*** a circle by its center later on, you must use the **Move tool**, since the **Selection tool** doesn't display the center of the circle when selecting it.

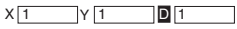
Center-Point Circle Tool

This tool draws a circle specified by the center point and diameter.

Using the Center-Point Circle tool

- Click two locations; the first click places the center and the second determines the radius, or
- ***Drag***, pressing at the center and releasing at a position indicating the radius.

If you ***drag*** the points, a *rubberband* image guides your construction. You can create a copy of the last circle by holding down the **Ctrl** key and clicking where you want the center for the new circle.



The status line shows the X, Y *coordinates* of the center and the *diameter* of the circle. **Diameter** is the selected status box.

Opposite-Point Circle Tool

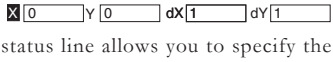
This tool draws a circle specified by the diameter.

Using the Opposite-Point Circle tool

- Click two locations to indicate the diameter.

  Or:
- ***Drag***, pressing the mouse button at the beginning and releasing it at a position that indicates the diameter.

If you ***drag*** the points, a *rubberband* image guides your construction.



The status line allows you to specify the **X**, **Y** *coordinates* representing the endpoints of the diameter.

3-Point Circle

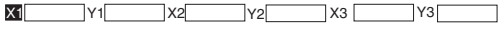
This tool draws a circle through the points you select.

Using the 3-Point Circle tool

1. Click the first point on the circle.
2. ***Drag*** or click the second and third points. (***Dragging*** displays the *rubberband* circle.)

If you place any of the three points on an existing object, the circle is drawn through that point. If you click an object while holding down the **Ctrl** key, the circle is drawn tangent to the object rather than through the indicated point. You may combine the placement of these points to create a circle through a specific point of one object and tangent to another object, or a circle tangent to three objects, or any other combination.

If you ***drag*** the first two points, a *rubberband* image guides your construction.



The status line indicates the X, Y *coordinates* for each of the three points.

Tangent-Point Circle Tool

This tool draws a circle tangent to the two objects you indicate.

Using the Tangent-Point Circle tool

1. Click the objects to which the circle is to be tangent.
2. You can enter a diameter for the circle in the status box.

Ellipse Tools

These tools on the tool palette construct ellipses inscribed within an invisible rectangle or parallelogram.

- The *2-Point Center* ***Ellipse*** uses the center point and one corner location of the rectangle.
- The *Opposite-Corner* ***Ellipse*** uses opposite corners of a rectangle.
- The *3-Point Center* ***Ellipse*** uses the center point, the midpoint of a side and the corner of the parallelogram.
- The *3-Corner* ***Ellipse*** uses three corners on a parallelogram.

Ellipses do not have ***center*** points by design. If you draw lines between opposite ***vertex*** points on the ellipse, then the ***intersection*** of the two lines will be the center of the ellipse. For a ***2 point Center*** *Ellipse* (inscribed in a rectangle) the Drawing Assistant's horizontal and vertical temporary construction lines will cross at the center point once you have touched two ***vertex*** points.

You can click or ***drag*** to create the ellipse. ***Dragging*** creates a rubberband ellipse, so you can see the ellipse before it is drawn.

The ellipse is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

2-Point Center Ellipse Tool

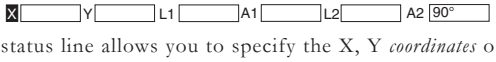
This tool constructs an ellipse inscribed within the rectangle calculated from the two specified points: the center point and one corner of the rectangle.

Using the 2-Point Center Ellipse tool

1. Indicate the center of the ellipse.
2. Indicate the corner of the rectangle defining the ellipse.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the **Ctrl** key and clicking where you want the center.



The status line allows you to specify the X, Y *coordinates* of the center point and the *length* and *angle* of the semimajor and semiminor axes of the ellipse.

Opposite-Corner Ellipse Tool

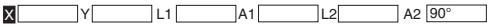
This tool draws an ellipse inscribed in the rectangle specified by the opposite corners you indicate.

Using the Opposite-Corner Ellipse tool

1. Indicate one corner of the rectangle defining the ellipse.
2. Indicate the opposite corner of the defining rectangle.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last ellipse by holding down the **Ctrl** key and clicking where you want the lower-left point.



The status line allows you to specify the X, Y *coordinates* of the lower-left point and the *length* and *angle* of the major and minor axes of the ellipse.

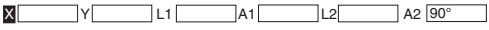
3-Point Center Ellipse Tool

This tool constructs an ellipse inscribed within the parallelogram calculated from three specified points: the center point, the midpoint of a side, and a corner of the parallelogram.

Using the 3-Point Center Ellipse tool

1. Indicate the center of the ellipse.
2. Indicate the midpoint of the side of the rectangle defining the ellipse.
3. Indicate the corner of the rectangle defining the ellipse.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the **Ctrl** key and clicking where you want the center.



The status line allows you to specify the X, Y *coordinates* of the center point and the *length* and *angle* of the sides of the parallelogram.

3-Corner Ellipse Tool

This tool draws an ellipse inscribed in the parallelogram calculated from the three corners you specify.

Using the 3-Corner Ellipse tool

1. Indicate one corner of the parallelogram defining the ellipse.
2. Indicate another corner of the defining parallelogram.
3. Indicate the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the **Ctrl** key and clicking where you want the lower-left corner of the parallelogram.

The status line allows you to specify the x, y *coordinates* of a corner and the *length* and *angle* of the sides of the parallelogram.

Polygon Tools

These tools on the tool palette draw rectangles, inscribed polygons or circumscribed polygons. Some of the rectangle tools create single lines which are individual objects or polygonal rectangles that can be selected and edited as a single object. ***Dragging*** to indicate the points of the polygon creates a rubberband image.

Use the entry boxes **D** and **Sides** in the status line to change any polygon when first creating it. After that, it becomes a series of separate line objects.

The polygon is drawn with the current pen specifications for *color*, *weight*, and *pattern*.

Rectangle Single Lines Tool

This tool draws a horizontal or vertical rectangle, using the opposite corners you specify.

Using the Rectangle tool

- Click the opposite corners of the rectangle.

  Or:

- ***Drag*** the opposite corners of the rectangle.

You cannot change the **Width** and the **Height** of a single lines rectangle in **Edit Objects**. You must edit each line individually, because rectangles become 4 line objects after you deselect the rectangle.

You can create a square by aligning the second point on the **45°** construction line. If the two points are on the vertical or horizontal axis, a straight line is drawn.

You can create a copy of the last rectangle by holding down the **Ctrl** key and clicking where you want the upper-left corner.

A default **1-inch** rectangle is drawn if you have not previously drawn a rectangle.

The status line allows you to specify the X, Y coordinates of the first point, as well as the width and height of the rectangle. **Width** is the selected status box.

If you want to draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides. You can't create a rectangle from the center.

Inscribed Polygon Tool

This tool draws a regular polygon where the radius of the circumscribing circle determines the location of the polygon's vertices. The default polygon is a hexagon, but you can specify the number of sides in the status line. (**Note**: The status line shows a diameter for the circle, the standard way of describing a polygon inscribed in a circle.)

Using the Inscribed Polygon tool

- Click the center of the polygon and a point on the circumference of the circumscribing circle.

  Or

- ***Drag*** from the center of the polygon to a point on the circumference of the circumscribing circle.

You can create a copy of the last inscribed polygon by holding down the **Ctrl** key and clicking where you want the center.

The status line allows you to specify the X, Y *coordinates* of the center, the *diameter* of the circle defining the polygon, and the number of *sides*. **Diameter** is the default status line selection, and the default number of sides is six.

Circumscribed Polygon Tool

This tool draws a regular polygon for which the radius of the circle determines the midpoint of the sides. The default shape is a hexagon, but you can specify the number of sides in the status line. (**Note:** The status line shows a diameter for the circle, the standard way of describing a polygon circumscribed around a circle.)

Using the Circumscribed Polygon tool

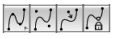
- Click the center of the polygon and the midpoint of one of the sides, or

- ***Drag*** from the ***center*** of the polygon to the ***midpoint*** of one of the sides.

You can create a copy of the last circumscribed polygon by holding down the **Ctrl** key and clicking where you want the center.

The status line allows you to specify the X, Y *coordinates* of the center, the *diameter* of the circle defining the polygon, and the number of *sides*. The default number of sides is **6**. **Diameter** is the default status line selection.

Spline Tools



The spline tools on the tool palette create **NURBs** (**N**on-**U**niform **R**ational **B**-Splines) which are a superset of Bezier curves. These splines are curves created by a complex mathematical formula.

NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact when the curve is changed. Kinks don't develop as the spline is altered. Second, NURB splines provide localized control of a complex curve. You can isolate an area and make changes without affecting the remainder of the spline.

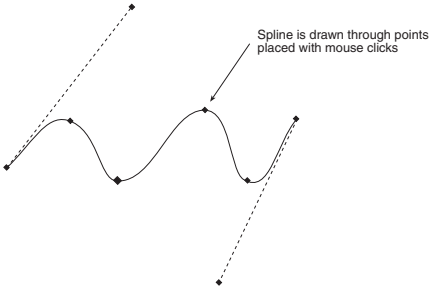
These properties are essential in aerodynamic designs. Air molecules moving over a wing surface must flow smoothly for maximum aerodynamic lift. If the surface does not maintain curvature continuity, the air molecules separate from the wing surface and cause a vacuum. Such a vacuum causes an eddy as the molecules try to fill it. This disruption of air flow increases the drag, which is not a part of an effective design.

The automotive industry also wants smooth air flow to improve gas mileage. Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete curvature continuity that makes a smooth reflection. Localized control of complex curves allows you to make minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, you could use a **NURB** spline to raise the center of the hood without changing the basic design.

NURB splines are also valuable for injection mold designs to eliminate the swirl of plastic as it is injected into the mold. Such designs provide better surface finishes and allow thinner cross-sections in the die.

Through-Points Spline Tool

The *Through-Points Spline* tool draws a spline through the points that you place.



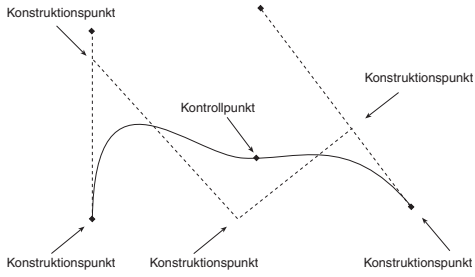
Using the Through-Points Spline tool

1. Click the points for the vectors of the spline.
2. Double-click the last point.

The status line shows the X, Y coordinates of each point as you go along.

Vector Spline Tool

This tool draws a spline using vectors determined by the points you specify. The **Vector Spline** tool uses each point that you place as the vertex of a vector for the spline it creates.



Using the Vector Spline tool

1. Click the points for the vectors of the spline.
2. Double-click the last point.

DraftBoard uses these vectors to calculate the control points (two fewer than the number of vertices you specify). The spline is tangent to the first and last vectors and passes through the calculated control points.

The status line shows the X, Y coordinates of each point as you go along.

Add Spline Control Point Tool

This tool allows you to add another control point to an existing spline.

Using the Add Spline Control Point tool

Click on the spline at the desired location(s) for the new control point(s).

***Note**: To see the new control point(s), as well as the existing control points, you must first select the spline and then choose **Show Points** from the **Layout** menu (or use the **Edit Objects** dialog box to specify **display points**).*

If you intend to move a point, be sure to lock the adjacent points so you don't change the slope of the rest of the spline.

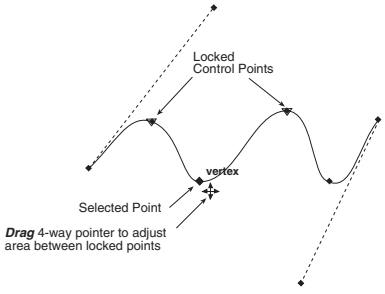
Lock Spline Control Point Tool

This tool allows you to lock the specified points on an existing spline so you can change the slope between the points without affecting the rest of the spline.

Using the Lock Spline Control Point tool

1. Choose **Show Points** from the **Layout** menu.
2. Click the ***vertices*** you want to lock.

The **Lock Spline Control Point** tool is used to lock or immobilize selected control points on an existing spline. You may then adjust the curvature of the spline between the locked points without disturbing other areas of the spline.



Unlocking spline control points

Use this tool and click the pointer to unlock it.

Locking more than one spline control point

You can lock more than one point by simply clicking each point.

Editing control points of a spline

1. Select the spline with the **Selection** tool.
2. Choose **Show Points** from the **Layout** menu. (If the menu says **Hide Points**, you don't need to choose it.)
3. Click on the drawing area to deselect the spline.
4. If necessary, select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. If necessary, click the points you want to lock (the points on either side of the point you will change).
6. Use the **Selection** tool to select the point you want to change.
7. ***Drag*** the point to the new location.

The dotted lines that appear tangent to the end portions of the spline are *tangent control arms*. They allow you to adjust the curvature of the spline leading up to the endpoints without moving the endpoints themselves. To do this, select the ***endpoint*** of the tangent control arm and ***drag*** it to another location with the **4-way pointer** that appears.

Editing the end slope of a spline

1. Select the spline with the **Selection** tool.
2. Choose **Show Points** from the **Layout** menu. (If the menu says **Hide Points**, you don't need to choose it.)
3. Click in the drawing area to deselect the spline.
4. If necessary, select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. If necessary, select the point you want to lock but not the endpoint.
6. Use the **Selection** tool to select the point at the end of the **tangent control arm** you want to move.
7. ***Drag*** the point to a new location.

CHAPTER

4

The Drawing Assistant

Snapping onto Geometry

Drawing Assistant Construction Lines

Permanent Construction Lines

The Drawing Assistant

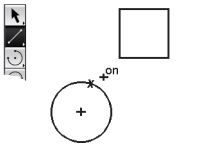
The *Drawing Assistant* is the feature that makes ***DraftBoard*** unique among design and drafting software products. The *Drawing Assistant thinks* like a drafter; it automatically *knows* where you typically want construction lines and displays them temporarily when you need them.

The *Drawing Assistant* also makes it easy to select existing points for construction by displaying information about the pointer's location in the drawing area. If a *Drawing Assistant* ***notation*** is displayed when you click, the construction snaps onto the geometry precisely, without requiring finely tuned, eye-hand coordination or tedious selection of special modifiers, modes, or other specialized construction tools.

Snapping onto Geometry

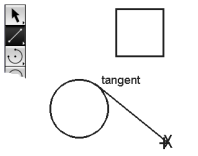
When the pointer is in the drawing area, it has a *snap point* function. The snap point locks onto specific points on existing objects as you move the pointer near them.

The *Drawing Assistant* tells you when the snap point is ***on*** an object.



The Drawing Assistant snaping onto a circle

The *Drawing Assistant* displays information about the location of the snap point. This information appears either beside the pointer or next to the object itself.



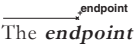
The Drawing Assistent displaying the relationship between the circle and the line that is being constructed

The *Drawing Assistant* tells you when the snap point has locked onto the points of an object.

center

The center of an arc or circle. Move the pointer across the arc or circle to display ***on*** for the arc or circle, then move the pointer near the center to display the ***center*** point notation.

endpoint

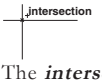


The ***endpoint*** of lines, arcs, circles, ellipses, and splines.

midpoint

The ***midpoint*** of lines, arcs, circles, ellipses, and splines.

intersection



The ***intersection*** of two lines or curves (both permanent lines and the *Drawing Assistant's* dynamic construction lines).

quadrant

Quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock, and 12 o'clock.

vertex

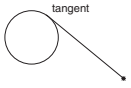
The ***vertices*** of an ellipse, spline, or dimension point.

Using tangents and perpendiculars

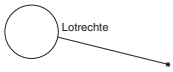
The *Drawing Assistant* locks onto a ***tangent*** or ***perpendicular*** only when the *Drawing Assistant* starts from the ***on*** notation. You cannot begin from a specific point, such as the ***endpoint***, ***quadrant*** or ***vertex*** point.

If you press a point on an arc or circle and ***drag*** the pointer away at about a 45° angle, the *Drawing Assistant* locks onto the ***tangent***. If you ***drag*** away at a 90° angle the *Drawing Assistant* locks onto a ***perpendicular***.

Tangent

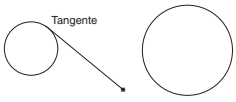


Perpendicular

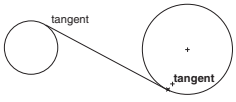


If you continue holding down the mouse button, the line remains ***tangent*** or ***perpendicular*** while you ***drag*** the ending point around the object.

This is a very useful feature if, for example, you want to create a line from and tangent to an existing circle to the tangent point of another circle.



Once a line is ***tangent*** to the circle, it can be ***dragged*** to the tangent point on the other circle, with the tangency maintained at both ends.



Keyboard Snap Points

Important: *Keyboard Snap Points are not active until you have set the first construction point of any drawing or editing tool.*

You can direct the *Drawing Assistant* to snap onto an object. For example, you may want to start a new line from the exact center of a circle. If you hold down the mouse button and press the **c** key on your keyboard, the *Drawing Assistant* finds the ***center*** of the circle when you move the pointer near the center.

Important: *Keyboard Snap Points are not active until you have set the first construction point of any drawing or editing tool.*

Using the Drawing Assistant for snapping onto geometry

Another way would be to reduce the detection radius of the *Drawing Assistant* with a lower **number of pixels** for the hit radius or to zoom in on the drawing to separate the construction points visually (see also *Hit Radius* in this chapter).

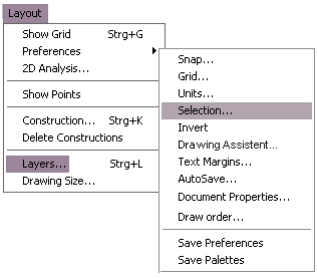
The following table lists the keys for finding specific points. The desired point must be within the hit radius of the pointer.

Letter	Snaps To (Align)
c	**c**enter
e	**e**ndpoint
g	**g**rid
i	**i**ntersection
m	**m**idpoint
n	**n**o point
o	**o**n
p	**p**erpendicular
q	**q**uadrant
t	**t**angent
v	**v**ertex
%	the percentage point set within the **Snap** dialog

Pressing the **Spacebar** or clicking the mouse releases all snap restrictions.

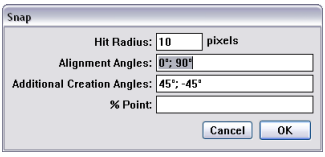
Drawing Assistant

This command under **Preferences** in the **Layout** menu allows you to turn the Drawing Assistant *on* or *off* and to activate the different *snap modes* of the Drawing Assistant.



Snap

This command in the **Preferences** submenu in the **Layout** menu sets specifications for the *Drawing Assistant*.



Hit Radius This setting determines the detection distance in pixels. When the pointer is within the specified hit radius, the *Drawing Assistant* notations are displayed and the object is selected when you click the mouse. The default hit radius is **12** pixels.

If you can't specify locations that are close together because the *Drawing Assistant* snaps to an existing *control point*, you can do any of the following:

- Decrease the hit radius.
- Zoom in so more pixels separate the existing point and the point you want to select.
- Lock on a point by pressing the mouse button and then typing the letter that represents the point (**m** for **m**idpoint, for example).

If you set the **Hit Radius** to zero, you disable the single click selection of the **Selection** tool (***dragging*** a marquee and double-clicking to select all objects will still work). Instead, you might consider using the **Selection Mask** in the **Edit** menu to specify that some objects cannot be selected.

Alignment Angles These angles define the *dynamic construction* lines that the *Drawing Assistant* automatically uses. If you want to change the orientation of your drawing, you can change these specifications. For example, you could set these angles to **30°**, **90°** and **150°** for an isometric drawing. The defaults are 0° (horizontal) and **90°** (vertical). Use a semicolon to separate the values.

To display a construction line through a point, move the pointer to the point to activate it (a diamond appears); construction lines automatically display through the active point. You can have as many as *eight* active points. When you activate the *ninth* point, the first one in the series is deactivated.

Additional Creation Angles These lines are used by the *Drawing Assistant* only when you are creating geometry and they are not part of the list of lines generated from the eight active points. The defaults are **45°** and **-45°**. Use a semicolon to separate the values.

% Point The divisions of a line for *Drawing Assistant* notations. If you want to divide the line into quarters, use a **25** specification. The default is **50**, which shows the ***midpoint*** of lines.

For example, entering *25* instructs the *Drawing Assistant* to tell you when the pointer is *25%* of the distance along a line as shown below.

Drawing Assistant Construction Lines

In addition to snapping onto geometry, the *Drawing Assistant* also displays *dynamic construction lines*. The three types of construction lines you use most frequently— vertical, horizontal, and **45°** angle lines— display automatically during construction. These construction lines appear temporarily to help you align geometry. Once you've set a point, the *Drawing Assistant* construction line disappears — your drawing is not cluttered with extraneous lines.

Dynamic construction lines extend automatically from the last point you created. You may want to activate other points so the *Drawing Assistant* displays construction lines relative to them. Simply move the pointer over the geometry to activate its control points, then move away horizontally or vertically.

Of course, you also have the option of creating *permanent construction lines* and other shapes, as described later in this chapter.

Using the Drawing Assistant's Construction Lines

In case you need different angles for these dynamic Construction lines, you can modify these angles in the **Snap** dialog box in the **Preferencres** submenu of the **Layout** menu.

After you have indicated the first endpoint of a line, and you move the pointer horizontally, vertically, or in a 45° direction, the *dynamic construction* lines will appear. The figure below illustrates a 45° construction line relative to the endpoint of an existing line.

Displaying dynamic construction lines while constructing geometry

1. Click a point to begin new geometry.
2. Move the pointer horizontally or vertically.

   A construction line appears through the point.
3. While the construction line is visible and the *Drawing Assistant* displays ***on***, click the next point.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly on that line when you clicked. The *dynamic construction* line disappears.

You can have as many as **eight** active points; activating the ninth point in a series deactivates the first point.

Displaying dynamic construction lines with existing geometry

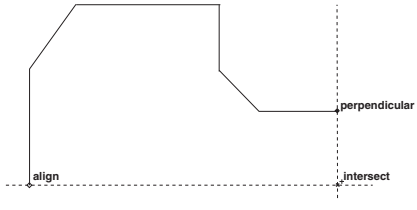
1. Without pressing the mouse button, move the pointer over an existing point.

   The point notation (***endpoint*** or ***midpoint***, for example) shows that the point is active.
2. Move the pointer away from the point horizontally, vertically, or at a **45°** angle.

   A construction line through the point is visible.
3. While the construction line is visible and the *Drawing Assistant* displays ***on***, click the desired point in the construction.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly ***on*** that line when you clicked. The *dynamic construction* line disappears.

The figure below illustrates intersecting construction lines drawn through two existing, active points.



Once you are familiar with the *Drawing Assistant*, you'll see how much ***DraftBoard*** streamlines design and drafting tasks!

Setting new Drawing Assistant construction angles

You can add to or change the angles that the *Drawing Assistant* uses for dynamic construction lines by choosing **Snap** from the **Preferences** command in the **Layout** menu. Enter the construction line angles, separated by semicolons, into the appropriate box.

Permanent Construction Lines

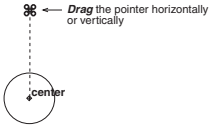
In addition to the *Drawing Assistant's* dynamic construction lines, you may want to create construction lines that remain displayed until you hide or remove them. There are two methods for creating *permanent construction lines*: strokes and the **Construction** command (in the **Layout** menu). When you want to get rid of all construction lines, choose **Delete Constructions** from the **Layout** menu. (Everything on the **Construction** layer will be deleted.)

Permanent construction lines automatically appear on the **Construction layer**, not the **work layer** of your drawing. You can hide the Construction layer to view or print the drawing without construction lines.

Stroke Construction Lines

Stroke construction lines are lines that you create with the mouse. Hold down the **Shift** and **Ctrl** key and ***drag*** the mouse horizontally or vertically. Since you are working in the drawing area, the *Drawing Assistant* helps you place the stroke precisely.

Holding down the **Shift** and **Ctrl** key changes the mouse pointer to the *Stroke* pointer (⌘).



You can use strokes to create construction lines while you are in the process of using a tool.

Drag	*Result*
Vertically	A vertical construction line through the first point of the stroke.
Horizontally	A horizontal construction line through the first point of the stroke.

Using Stroke construction lines

Construction lines are as long as the dimensions of the viewing area of the screen or the plot region (as designated in **Drawing Size**), whichever is larger.

Stroke construction lines can be created only *horizontal* or *vertical*. For *construction lines* in a different angle you must use the **Construction** command in the **Layout** menu.

For example, if you are using the **Connected Line** tool, you can create a construction line that extends through the center of a circle:

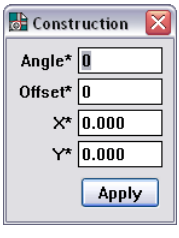
1. Hold down the **Shift** and **Ctrl** key.
2. Move the pointer near the center of the circle.
   The *Drawing Assistant* snaps onto the center point.
3. ***Drag*** the mouse vertically or horizontally away from the ***midpoint***.
   The construction line appears through the ***center*** and you are still in the process of creating connected lines.

The Construction Command

Stroke construction lines are useful for creating lines through existing points. If you want to create a construction line at a location other than an existing point or at a particular angle, you can use the **Construction** command.

Construction Ctrl+K

This command in the **Layout** menu creates a construction line on the **Construction** layer of the current document.



DraftBoard automatically places construction lines on the **Construction** layer. When you choose **Delete Constructions** from the **Layout** menu, **everything** on the **Construction** layer is deleted, regardless of the object's pen style.

You can specify the angle of the construction line or the offset from a reference point defined by the X, Y coordinates. The asterisk shows that the values can be specified by clicking or dragging the mouse; values can also be typed into the boxes.

Specifying the construction line angle with the mouse

1. Click the **Angle** box.
2. ***Drag*** a vector in the drawing area.

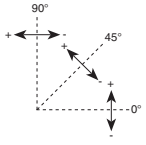
   The angle of the vector line appears in the **Angle** box.

Specifying the construction line offset with the mouse

1. Click the **Offset** box.
2. ***Drag*** the offset distance in the drawing area.

*Note: The distance **dragged** will always be entered in the **Offset** box as a positive value, regardless of the direction **dragged**.*

The distance you ***drag*** appears in the **Offset** box. Be aware that the offset is determined by the angle of the construction line as shown.



Specifying X, Y coordinates with the mouse

The coordinates of the last point you specified appear in the **X** and **Y** boxes, but you can change them by following these instructions.

1. Click the **X** box.
2. Enter new coordinates.

   Or:

   In the drawing area, click the location of the point through which the construction line should pass. The coordinates are entered automatically for both the **X** and **Y** boxes.

*Before you can define a point on the drawing area you have to click in the **Entry field** where the values should go.*

Creating multiple construction lines

You can create multiple construction lines through the same point by entering the angles you want separated by semicolons.

Creating parallel construction lines

You can create parallel construction lines by specifying a single angle value with different offsets separated by semicolons. You can also create parallel lines by creating one construction line using this box, then creating new lines with the **Parallel Line** tool.

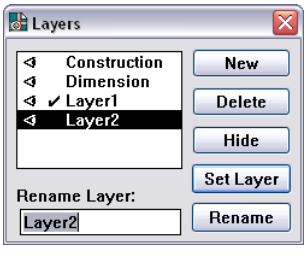
Creating Construction Geometry

*Lines made with the **Construction** pen do **not** go on the **Construction** layer unless you make that layer the work layer.*

Non-construction geometry is placed on the work layer. You can create temporary construction geometry, such as arcs or circles, by making the **Construction** layer the work layer, creating the geometry and switching to another layer to continue your work. You can use the **Construction** pen style if you like, but it's not essential.

Creating construction geometry

1. Choose **Layers** from the **Layout** menu.



2. Click **Construction** in the list box.
3. Click the **Set Work** button.
4. Create the geometry you'll use for construction. You can use the **Construction** pen style, but any pen style is acceptable.

*Since the **Delete Constructions** command deletes all geometry on the **Construction** layer, you shouldn't create anything you want to keep on the **Construction** layer.*

5. When the construction geometry is complete, make another layer the work layer.
6. Close the dialog box.
7. Continue your work.

Once you no longer need the construction geometry, choose **Delete Constructions** from the **Layout** menu to remove all geometry on the **Construction** layer.

Removing Construction Lines

If you have many construction lines and you want to delete only one or two of them, select the lines you want to remove, and then choose the **Delete** command from the **Edit** menu or press the **Back Space** key. You can remove all the construction lines that you've created by choosing **Delete Constructions** from the **Layout** menu.

You can select construction lines in the usual manner from any layer; the **Construction** layer doesn't have to be the work layer.

Delete Constructions

This command in the **Layout** menu deletes all construction lines and any geometry on the **Construction** layer.

The Drawing Assistant's *dynamic construction* lines appear only temporarily and are not affected by this command. Any geometry on the **Construction** layer (regardless of the pen style used) is deleted by this command.

You can retrieve deleted construction geometry within the limits of the **Undo** command.

CHAPTER

5

Setting the Environment

Pens

Setting the Units

Displaying the Grid

Preferences

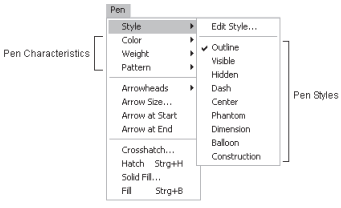
Setting the Environment

This chapter describes how to set pen styles, units and discusses the grid. The last section of this chapter tells you how to save your preferences for default settings.

Pens

The pen style determines the appearance of lines on the screen and when they are plotted. If you are using a monochrome monitor or a printer, all lines will be black but the weight and pattern will be visible. Any line thickness of less than **0,07 mm** appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

You can choose from nine different pen styles in the **Pen** menu. This menu contains as well all commands to modify all pen settings.



The current pen style is checked on the **Pen** menu. The default pen style is *Outline—solid, black lines, **0,35 mm** wide.*

You can change to a different pen style in the **Style** submenu.

You can change an individual characteristic of the current pen in the **Pen** submenus (**Color**, **Weight** and **Pattern**).

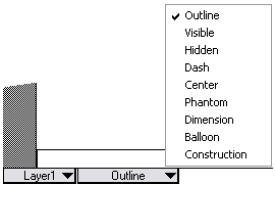
Changing the current Pen

1. Select the pen style you want to use from the **Style** submenu in the **Pen** menu.

All selected objects will be drawn in the new pen style, as will all future lines until you end the current ***DraftBoard*** session or select a new pen style.

Pen Indicator

You can also use the *Pen Indication **pop-up** box* at the lower left of the drawing area to specify the current pen.

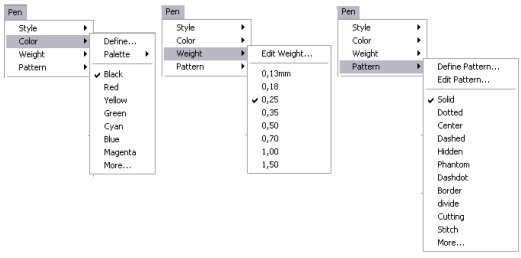


Changing the current pen with the Pen Indicator

1. Press the mouse button on the Pen Indicator box.
   The **Pens** menu displays listing all specified *Pen Styles*.
2. ***Drag*** to the *Pen Style* you want to make the current pen and release the mouse button.
   The selected pen is activated and displayed in the *Pen Indicator*.

Pen Characteristics

The following characteristics are available from the submenus in the **Pen** menu.



If you want to change the characteristic of a pen permanently use the **Edit Style** command from the **Style** submenu in the **Pen** menu. This command is described later in this chapter.

Whenever you change the characteristic (*Color*, *Weight* or *Pattern*) of a pen these characteristic are valid until you end the current ***DraftBoard*** session or select a new pen characteristic.

Changing one characteristic of a pen style

1. Choose the pen you want to modify.
2. Choose a characteristic—*color*, *weight*, or *pattern* – from the characteristic submenus in the **Pen** menu.

To return a standard pen to its original specifications, choose a different pen style from the menu and then choose the modified pen again.

All selected objects will reflect the new pen characteristic, as will all future lines drawn until you end the current ***DraftBoard*** session or select new pen characteristics .

Changing the pen characteristics of an object without changing the pen

1. Select the object.
2. Choose **Edit Objects** from the **Edit** menu.
   The **Edit Objects** dialog is displayed.
3. Change the pen characteristic in the **Edit Objects** dialog box.
4. Click the **Apply** button.
   The selected objects reflects the new pen characteristics. The characteristics of the current pen are not modified.

Colors

The **Color** submenu in the **Pen** menu contains **234** color options from a palette of *16.7 million* colors available. **64** colors are predefined, **170** colors are user definable.

You select a new color for the current pen from the **Color** submenu in the **Pen** menu where you find also the command to define your own colors.

Color

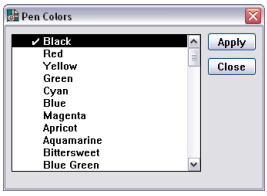
This command in the **Pen** menu specifies the **color** of the current pen and any selected geometry without changing any other pen characteristics.

You can change the color of the current pen in the **Color** submenu by selecting a *color name* or by selecting a new color from the *Color subpalette*.

Specifying the color of the current pen by selecting a color name

1. Select the **Color** command from the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the color you want.
   The current pen and all selected objects takes on the new color and the color is selected in the submenu.

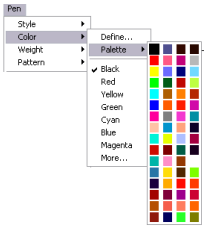
3. Select **more** from the **Color** submenu, if you want to select a different color than one of the seven displayed basic colors.
   The **Color** dialog box is displayed listing all available colors.



4. Choose a color name.
5. Click the **Apply** button
   The current pen and all selected objects takes on the new color.
6. Click the **Close** button when you have modified all line colors.

Choosing the color of the current pen from the color palette

1. Select the **Color** command from the **Pen** menu.
   The submenu appears.
2. ***Drag*** to **Palette** in the **Color** submenu.
   The **Color Palette** is displayed showing 64 predefined and all user defined colors.



3. ***Drag*** to the color you want.
   The current pen and all selected objects takes on the new color and the color is selected in the submenu.

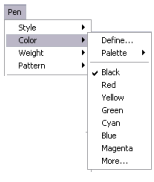
Changing the pen color of an object without changing the color of the current pen

1. Select the object.
2. Choose **Edit Objects** from the **Edit** menu.
   The **Edit Objects** dialog is displayed.
3. Change the pen color in the **Edit Objects** dialog box.
4. Click the **Apply** button.
   The selected objects reflects the new pen color. The color of the current pen is not modified.

Define Colors

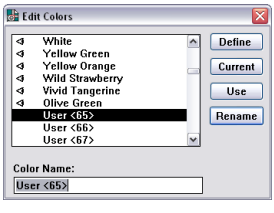
You can define **170** colors in addition to the **64** predefined colors.

New colors are defined with the **Define** command from the **Color** submenu in the **Pen** menu.



Specifying a new color name

1. Choose the **Define** command from the **Color** submenu.
   The **Define Color** dialog box is displayed.



The **Define Color** dialog box displays a list containing all color names that can be defined in ***DraftBoard***. The list starts with the *64 predefined* colors followed by the *170 user definable* colors named **Color <65>** to **Color <236>**.

2. Scroll down the list and select the first undefined color number, like **Color <65>**.
3. Enter the color name (limited to 15 characters) in the **Color Name** data field.
4. Click the **Rename** button.

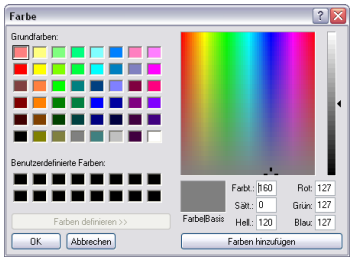
   The undefined **Color <65>** is renamed.

Now we specify a color for the new color name.

Defining a color

1. Click the **Define** button in the **Define Color** dialog box.

   The *Windows* ***Color*** *dialog box* is displayed.



2. Click the **Define Custom Color** button.

   The extended **Color Palette** is displayed.
3. In the color palette on the right, place the cursor within the target cursor and ***drag*** to the area of the color desired. Note a larger area of the color is displayed in the **Color/Solid** box below.

   If desired, you can also adjust the *hue*, *saturation*, *luminance*, and the percentage of *red*, *green*, and *blue* by entering your own values.

   The % of black can be adjusted by moving the cursor over the arrow in the color scroll bar to the right of the large color palette and drag up or down to the desired location. When you adjust this, all the values in *Luminance*, *Red*, *Green*, and *Blue* data fields change as well. *Hue* and *Saturation* do not change.
4. Click the **Add to Custom Colors** button.

   The defined color is displayed in the first highlighted box for custom defined colors.
5. Click the **OK** button.

   Die *Windows* ***Color*** dialog box closes and you are returned to the **Define Colors** dialog box.
6. Click the **Use** button in the **Define Colors** dialog box.

   An *icon of an eye* will appear, indicating that the color will be visible now in the ***DraftBoard*** *Color Palette*. The new color name remains in the color list in the position you placed it, since the color names are not alphabetized.
7. Close the **Define Color** dialog box.

Saving user defined colors

User defined colors are automatically saved with the drawing as soon as you save the drawing using the **Save** command from the **File** menu.

To save user defined colors permanently choose the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Arranging the color palette

All *user defined colors* are automatically displayed in the ***DraftBoard*** *Color Palette* when an *eye symbol* is displayed in front of the color name in the **Define Color** dialog box.

If there are any undefined colors between the last *predefined color* (**Color <64>**) and the last *user defined color* all these undefined colors are shown in the color palette by a cross.

If you want to have a continuous color palette you must define all color continuously after the last predefined color.

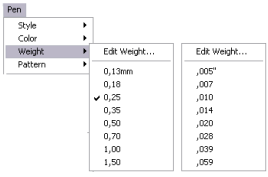
Pen Weights

The **Weight** submenu in the **Pen** menu offers **8** pen weights, that are all user definable. This submenu contains also the **Define** command, that allows you to specify for each of the pen weighs displayed in that menu a specific weight.

Weight

This command in the **Pen** menu sets the pen **width** of the current pen and selected lines without changing any other pen characteristics.

The line weights shown in this submenu are dependent on the selected **Units** in the **Preference** Submenu. If you have selected metric units, the line weight will be shown in **mm** instead of **inches**.



Specifying a new weight for the current pen

1. ***Drag*** to **Weight** in the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the weight you want.
   The pen takes on the new weight and the weight is selected in the submenu.

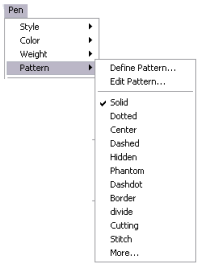
Any line thickness of less than **0,07 mm** appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights. The *Line weight* of *dimension lines* can be only modified in the **Preferences** submenu of the **Dimension** menu and should be always **0,25 mm**.

Line Patterns

DraftBoard Unlimited allows to specify **19** line patterns in addition to **11** predefined line patterns.

Pattern

This command in the **Pen** menu sets the **Pattern** of the current pen and any selected lines without changing any other pen characteristics.



The **Pattern** submenu in the **Pen** menu contains all defined line patterns. **11** different line patterns can be selected directly, for all others you have to display a dialog box using the **More** command.

Specifying line patterns in the Pattern submenu

1. ***Drag*** to **Pattern** in the **Pen** menu.
   The submenu appears.
2. ***Drag*** to the pattern you want.
   All selected objects take the new pattern and all new objects are drawn in this line pattern.

Editing Line Patterns

With the **Edit Pattern** command from the **Pattern** submenu in the **Pen** menu you can set the spacing for ***DraftBoard's*** line patterns. ***DraftBoard*** automatically scales line patterns by the inverse of the scale factor set in **Drawing Size** or **Sheet Into View** so that the pattern spacing will be appropriate for the viewing and drawing scale of the geometry. Sometimes, though, you might want to set the pattern spacing independently of the viewing or drawing scale, or change the pattern altogether.

The **Edit Pattern** dialog box lets you control the pattern spacing for line patterns.

In ***DraftBoard*** you can edit all line patterns except **2** (*solid* and *dotted*) of the **11** predefined line patterns.

That means that changing, for instance, the **Phantom** line pattern in the **Edit Pattern** dialog box will change the pattern of every line in the existing document that is drawn in the **Phantom** line pattern.

All line patterns are edited in the *Line Editor* that you display using the **Define Patterns** command in the **Pattern** submenu of the **Pen** menu.

Pen Styles

The following styles are the default settings of those pens that are available in the **Pen** menu:

Outline

Pattern: **Solid**, Weight: **0.35 mm**, Color: **Black**. This is the default pen setting.

Visible

Pattern: **Solid**, Weight: **0.5 mm**, Color: **Black**.

Hidden

Pattern: **Hidden**, Weight: **0.35 mm**, Color: **Red**. Hidden lines are actually visible on the screen and in your drawing; they represent lines that would be hidden in a solid object.

Dash

Pattern: **Dashed**, Weight: **0.35 mm**, Color: **Yellow**.

Center

Pattern: **Center**, Weight: **0.35 mm**, Color: **Green**.

Phantom

Pattern: **Phantom**, Weight: **0.35 mm**, Color: **Cyan**.

Dimension

Pattern: **Solid**, Weight: **0.25 mm**, Color: **Blue**. Dimension lines are used for all type of dimension.

Balloon

Pattern: **Solid**, Weight: **0.25 mm**, Color: **Green**.

Construction

Pattern: **Dotted**, Weight: **0.25 mm**, Color: **Magenta**.

The *Construction* line style doesn't create *dynamic* or *permanent Construction Lines*. This line style uses the same line pattern but is not automatically placed on the **Construction** layer and can't be deleted with the **Delete Constructions** command from the **Layout** menu.

Modifying Pen Styles

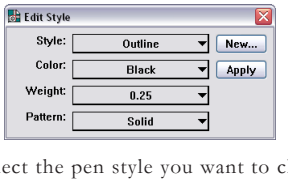
You can change the characteristics of a pen the **Edit Style** command in the **Pen** menu.

Edit Style

This command in the **Pen** menu sets the characteristics for the pen styles.

Redefining the specification of a pen style

1. Choose **Edit Style** in the **Pen** menu.
   The **Edit Style** dialog box appears.



2. Select the pen style you want to change by pressing the **Style** entry box. When the list of pen styles appears, ***drag*** to the style you want to change.
3. Specify the characteristics (*color*, *weight*, and *pattern*) you want for that style.
4. Click the **Apply** button.
   The selected pen style has now the new specifications until you quit ***DraftBoard***. When you click the **Apply** button all selected objects in the drawing area obtain the new specifications. The dialog box stays open so you can make changes to other pen styles.

To save *Pen Styles* permanently you must choose the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu. See *Saving Preferences* later in this chapter.

When you edit a style, you set new specifications for future uses of that pen. The specifications remain in effect for the document in

which they are set until you change it again with the **Edit Style** command.

Redefining the specification of a single pen style

When you only want to modify the characteristics of a pen as long as you use this pen you must proceed as follows:

1. Select the pen style you want to change
2. Specify the characteristics (*color*, *weight*, and *pattern*) you want for that style.

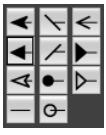
When you want to restore the original characteristics of the pen you select any other pen and then again the pen whose attributes you have modified.

Arrow Lines

With the commands **Arrow at Start** and **Arrow at End** from the **Pen** menu you can specify the placement of *Arrowheads* on lines or circular arcs that are not a part of dimensions. You can choose the Type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. The size of the arrowhead can be specified in the **Arrow size** submenu of the **Pen** menu.

Arrowheads

The **Arrowheads** submenu of the **Pen** menu allows you to choose one of eight arrowheads for arrow lines.



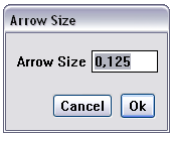
Arrowheads for *dimension lines* are defined separately under **Preferences** in the **Dimension** menu.

The arrowhead selection is valid until the end of the current ***DraftBoard*** session. You can save the selected *Arrowhead* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Arrow size

The size of an arrowhead is specified in the **Arrow size** submenu of the **Pen** menu. Choosing this command displays the following dialog box:

The size of dimension arrowheads is set separately under **Preferences** in the **Dimension** menu.

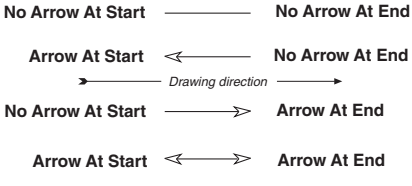


For the arrow size you can enter a factor in the **Arrow size** entry box. The default factor is **3,5**. A modified arrow size is valid until the end of the current ***DraftBoard*** session.

To save the size of an *Arrowhead* permanently you must choose the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Arrow at Start

This command in the **Pen** menu places an arrowhead at the beginning of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. A checkmark (✓) indicates the current arrowhead setting.



Arrow at End

This command in the **Pen** menu places an arrowhead at the end of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the **Arrowheads** submenu of the **Pen** menu. A check mark indicates the current arrowhead setting.

Creating arrow lines or arcs

1. Select one or several lines you want to place an arrowhead at the beginning or at the end.
2. Choose an arrowhead from the **Arrowhead** submenu of the **Pen** menu.
3. Select **Arrow at Start** or **Arrow at End** in the **Pen** menu. If you select both commands arrowheads are placed at both ends of the selected lines or arcs.
4. Use the **Arrow size** command of the **Pen** menu to adjust the size of the placed arrowheads.

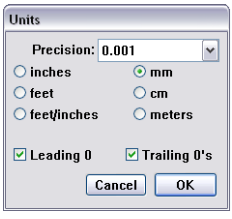
Setting Units

DraftBoard is set to measure geometry in millimeters. When you open a new drawing you should set the precision, units, and fractional or decimal specifications, to suit your needs.

Units

This command is in the **Preferences** submenu in the **Layout** menu.

When you change the units, existing dimensions update to reflect the change, converting automatically between English and metric.



Altering the precision does not affect any dimensions. The precision of dimensions is selected in the **Preferences** submenu of the **Dimension** menu.

Precision Sets the number of decimal places in all data fields within ***DraftBoard*** (in *Edit Objects*, the *Status line*, and all *Dialog boxes*). The default is **0,01**.

When you choose a **fractional** precision, measurements will appear as fractions rather than decimals. You can set the format for fractions in the **Linear** submenu in the **Dimension** menu.

When ***DraftBoard*** rounds off values it stores the 16-digit decimal number; therefore, you can change the precision displayed at any time and maintain accuracy.

Inches, feet Displays measurements in English units. Measurements less than *6 feet* appear in inches and those greater than *6 feet* appear in feet and inches.

mm, cm, meters Displays measurements in metric units.

The options **Leading 0** and **Trailing 0's** set only the number of decimal places in all data fields (*Edit Objects, Status Line, Position Indicator*) and in dialog boxes like **Edit Objects**, **2D Analysis** etc.). Leading **O** and **Trailing 0's** for dimensions are set in the **Preferences** submenu of the **Dimension** menu.

Leading 0 Determines whether or not the zero to the left of the decimal point appears.

Trailing 0's Determines whether or not zeros to the right of the decimal point are displayed. If set, the precision determines the number of trailing zeros.

Using the status line to specify the unit of measure

You can use the status line to specify a unit of measure other than the one in effect for the document. If the units are set to inches and you draw a line, you can use different units of measure such as inches ("), feet ('), feet and inches (x'y"), millimeters (mm), centimeters (cm), and meters (m). You can also mix the units in the mathematical expression as long as you label the units properly-for example, 10" + 25.4 cm.

In the following example, if the units are set to **Meter**, and a line is drawn using the value below, a line **23 cm** long will be drawn when you press the **Enter** key, and the status line will show the measurement in meters.

In addition, you can use *mathematical*, *exponential*, and *trigonometric* expressions in the status line.

Displaying the Grid

When the grid is visible, objects snap to the grid spacing. If you want to place an object between the grid marks, you must either turn off the grid, change the grid spacing, or zoom in so that the spacing is larger than the hit radius. The grid spacing automatically reflects the units you set in the **Units** dialog box. When you change the units, the appearance of the grid spacing remains the same, and the values listed in this dialog box change automatically.

Grid

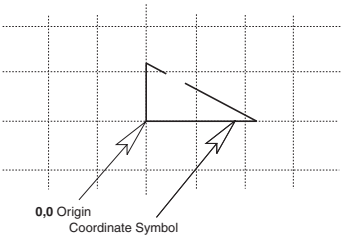
This command in the **Preferences** submenu on the **Layout** menu allows you to set the spacing of the grid lines. You can specify the number of ticks (**subdivisions**) per unit (**spacing**). You can change the default setting by saving changes in the preferences file.

When you make any changes in the **Grid** dialog box, the grid is automatically displayed after closing the **Grid** dialog box.

Show Grid Ctrl+G

This command on the **Layout** menu controls the display of the grid. The grid is made up of horizontal and vertical lines of dots. The spacing of the grid lines and the number of tick marks between the intersections are set in the **Grid** dialog box which you can choose from the **Preferences** command on the **Layout** menu.

When the grid is visible, the *Drawing Assistant* snaps to the grid spacing. In other words, if the grid is set to *25 mm* spacing, you can't construct an object closer than *25 mm* to another object. The coordinate symbol appears at the origin location (**X=0, Y=0**) when the grid is visible.



Displaying the *Grid* is rarely required, since the *Drawing Assistant* offers in most cases a better support than the *Grid*.

Saving Preferences

All files are saved with the settings as they are established when you save and close the file. The characteristics used for new files (the default settings) are contained in the *prefs.vlm*. The file must be stored in the same directory as the ***DraftBoard*** application.

Preferences

This command in the **Layout** menu displays a submenu for setting various specifications. Descriptions of the items in the submenu appear individually in this reference section. You can change the default setting by saving changes in the preferences file, ***prefs.vlm***.

Snap Use this dialog box to set the alignment angles, hit radius, and other settings for the Drawing Assistant.

Grid This dialog box allows you to specify the spacing of the grid lines. Use the **Show/Hide Grid** command in the **Layout** menu to turn the display of the grid on and off.

Units This dialog box specifies the units and precision for the current ***DraftBoard*** document.

Selection Use this dialog box to set the color or blinking which shows that an object is selected.

Invert This commands changes the *Background color* (the color of the drawing area from *white* to *black*)

Text Margins In this dialog box you can specify values for the *Left*, *Right*, *Top* and *Bottom* text margins.

AutoSave This dialog box allows the user to set a time interval for the program to save files.

Document Properties	In this dialog box the user can set document properties such as *Author*, *User defined attributes* or to automatically create a document *Preview*.
Drawing Assistant	In this dialog box you can turn the *Drawing Assistant On* or *Off* and activate different *snap modes*.
Draw Order	This command activates the **Draw Order** that will display all objects according to the **Bring to Front/ Bring to Back** command when redrawing the screen. In complex drawings switching this command off can accelerate the screen redraw.
Save Preferences	If you have set the characteristics you like, such as units, pen style and dimension format, you can make those settings the default setting. Choose **Save Preferences** from the **Preferences** submenu on the **Layout** menu. You cannot save geometry in this way; you would have to open the preferences file, create the geometry, and save the file.
Save Palettes	When you select this command the *Status* (visible/ hidden) and the *Position* of the *Dimension palette* is saved to preferences. When you restart ***DraftBoard*** the *Dimension palette* if it was displayed when you closed ***DraftBoard*** will be automatically reopened. *Status* and *Position* of the *Dimension palette* is saved in the **DraftBoard.ini** file under the section **Palettes**.

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If a palette is not visible it could be that it was saved at a position outside of the screen. In this case you look for the name of the palette in the *DraftBoard.ini* file in the ***DraftBoard*** directory or folder under the section Palettes and enter there the Pixel Coordinates 20, 20. Then you must restart ***DraftBoard***.

Preference File *Prefs.vlm*

The preferences file name is *prefs.vlm*. The file must be stored in the same directory as the ***DraftBoard*** application.

If you want to use different settings for your work, you can change the default settings, and then every new document opens with the settings you have selected. The following specifications can be set in the preferences file:

- Pen styles
- Text characteristics
- Text Margins
- Preferences settings (*Snap*, *Grid*, *Units*, and *Selection* indicator, *Auto Save*, *Document Properties* and *Directory* settings)
- Grid display
- Layer specifications
- Work layer
- Dimension and tolerance formats
- Arrowhead type and display
- Drawing size and scale
- Zoom scale
- Fillet radius
- Chamfer angle and length
- User Defined Colors

You cannot save geometry into the preferences file with the **Save Preferences** command, only your settings are saved. Be careful to use the **Save Preferences** command when your drawing contains several *Detail Views* and *Layers*. Although this command doesn't save any geometry, it saves the setup of the current drawing as default for all future drawings you create. That means that any new drawing will contain the same number of *Detail Views* as the drawing you used the **Save Preferences** command with. Therefore check regularly your *Preference File*. It should contain per default ***no*** *Detail View* and ***4*** *Layers* (Layer 1, Construction, Kinematic and Dimension).

Changing the default settings

1. Create a file which has the preferences you want.
2. Choose **Save Preferences** from the **Preferences** submenu in the **Layout** menu.
   The preferences are set for subsequent new documents.

CHAPTER

6

Selecting Objects

Objects

Indicating Selection

Selection Process

Selecting Objects

Once you have constructed objects, you may want to make changes. One of the basic rules of ***DraftBoard*** says that you must first select an object before you can edit it. Just as you first have to select a drawing tool from the **Tool palette** before you can start drawing, an object must first be selected before a function can be applied.

Modifying an object is always a two-step process:

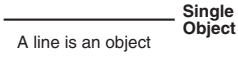
- Select the object.
- Specify the action for the selected object.

For example, you could select a circle and then change the pen style to **Center** to indicate a bolt-hole circle.

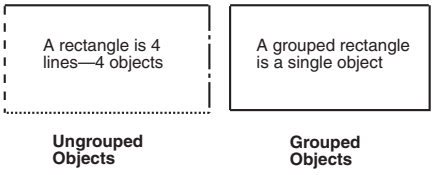
This chapter deals with selecting both *objects* and *points*, since they behave in different ways.

Objects

A single piece of geometry (for example a point or a line) is an object.

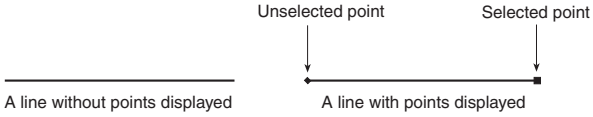


Several objects that have been grouped with the **Group** command are also an object. For example, the four lines of a *Single Line* rectangle are four objects. If you group them, ***DraftBoard*** treats them as a single object.



A point is an object, too. Every type of geometry contains two or more points, sometimes called *control points*. A line has two control points, one at the beginning and one at the end of the line. If you select an object and choose **Show Points** from the **Layout** menu, the points are visible on the selected object.

The *Starting* and *Ending point* of a circle is displayed by the *Drawing Assistant* as ***Vertex***.



If you select a point without selecting the geometry it defines, ***DraftBoard*** treats the point as an object.

Point selection is described in a section later in this chapter.

Note*: Keep in mind you need to select an object and than the function.*

Indicating Selection

When you select an object, its appearance on the screen shows that it is selected, it is a specific color (such as red), it blinks, or both. ***DraftBoard*** usually displays selections according to your monitor type, red for a color monitor and blinking for a monochrome monitor. You can change the color or turn blinking off and on for the selection indicator. You can choose both color and blinking, if you want.

Selection

This command in the **Preferences** submenu in the **Layout** menu determines the appearance of selected objects. If you have a color monitor, selected objects appear in a **Color** (which you can change with this setting). If you have a monochrome monitor, selected objects **blink**.

If you have a color monitor, you can also specify blinking with or without color.

Indicating the Selection

You shouldn't use black for indicating selection because many of the standard pens use black lines. Red is a good choice unless you are color blind, in which case blue may be a better choice.

1. Choose the **Selection** command from the **Preferences** submenu in the **Layout** menu.

   The **Selection** dialog box is displayed.

2. Select the option **Blinking** and/or **Color**.

   To choose a color, move the pointer to the color box and click into the box to display the choices. ***Drag*** to the color you want and release or click the color.

   It is not recommend to use the color **Black** to indicate the selection, since many lines are displayed in black.

3. Click the **OK** button.

   Once the selection color is changed, all current and future selected items appear in the new color until you quit ***DraftBoard***.

   Using the **Save Preferences** command you can save the new settings permanently.

Selection Process

Selecting an object does not affect the properties of the object. A selected object is highlighted, but this highlighting goes away once the object is deselected. While points and objects are selected in a similar way, point selection is controlled by the **Selectable Points** setting in the **Edit** menu. For this reason, selecting objects and selecting points are discussed separately.

Selection Tools

The **Selection** tool palette contains all tools for selecting objects.

Objects can be selected by *clicking*, by ***dragging*** a *Selection Marque* around a group of objects, by ***dragging*** a *Line* over objects or by drawing a *Selection Polygon* around objects. In the **Selection** tool palette you find as well the **Tracer** tool for automatically selecting boundary lines.

All **Selection** tools allow when pressing the **Shift** key to select or deselect additional objects.

Selection Tool

The **Selection** tool on the tool palette selects one or more objects or points in the drawing area.

Selecting a single object

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.

The object is selected, and previously selected objects are deselected.

Selecting multiple objects by shift-clicking

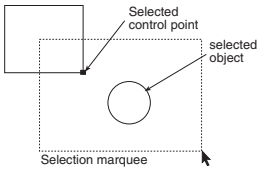
1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.
3. Press and hold down the **Shift** key.
4. While holding down the **Shift** key, click other objects to be selected.

   The objects you clicked are selected. If you then click one of the selected objects, it is deselected.
5. Release the **Shift** key.

Selecting multiple objects by *dragging*

If you want to select more than one object, you can ***drag*** with the mouse button pressed a selection marquee around the objects.

1. Click the **Selection** tool in the tool palette.
2. ***Drag*** a selection marquee around the objects you want to select.



All objects that lie completely inside the selection marquee are selected. If a portion of an object is outside the region you ***dragged***, that geometry is not selected but the *control points* of that geometry which lie inside the selection marquee are selected.

If you want to select most of the objects within an area, you can ***drag*** a selection marquee to select all the objects, and then deselect the objects you do not want selected by holding down the **Shift** key and clicking them.

Select all Objects

Double-clicking the **Selection** tool selects all objects on the drawing area.

Tracer

This tool on the **Selection tool** subpalette selects automatically boundary lines. The tool uses coincident endpoints and line intersections to trace the perimeter of the geometry.

The **Tracer tool** is most useful for selecting boundaries for *crosshatching*, for *solid fills* and for *2D Analysis* because it eliminates the need to segment geometry.

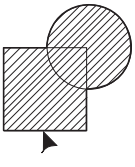
Selecting boundary lines

1. Select the **Tracer tool** from the **Selection Tool** subpalette.
2. Click on a boundary line.

   The tracer moves from one line to another making its selection.

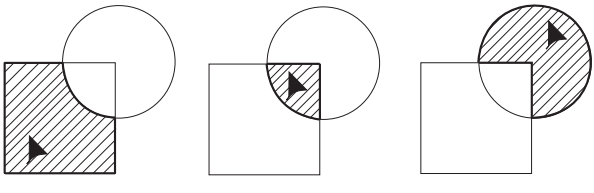
***Important**: You can press the **Escape** key to stop the **Tracer**.*

If you have two overlapping objects as shown on next page, the tracer selects the perimeter of the objects, depending on where you click.



If you click on the outside of the perimeter line, the **Tracer** selects the outer most perimeter. If you click on the inside of the line, the **Tracer** selects the inner most perimeter.

The crosshatching on the figures below shows the boundaries selected when you click the **Tracer** tool at the cursor locations shown.



Intersecting areas

If you want to determine and isolate the overlapping part of two intersecting areas you proceed as follows:

The **Tracer tool** creates a *temporary* copy of the boundaries. Temporary because it's copied to a *temporary* layer, which is deleted as soon you activate the **Tracer tool** or select a command other than **Hatch** or **2D-Analysis**. As long as the *temporary* boundaries are selected they can be copied to another layer.

1. Select the **Tracer tool** from the **Selection Tool** subpalette.
2. Click on the overlapping area.
   The overlapping area is selected.
3. Select **Layer** in the **Layout** menu.
4. Create a new layer.
5. Select **Edit Objects** in the **Layout** menu.
6. Select the new layer in the **Layer** list box and click the **Apply** button
7. Close the **Edit Objects** dialog box.
8. Make the new layer in the **Layer** dialog box the work layer.
9. Hide the layer that contains the overlapping object geometry.

The new layer contains now the intersecting area which you can edit like any other object.

Selection commands

The **Edit** menu contains the two selection commands **Select All** and **Selection Mask**.

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Select All Ctrl+A

Select All is useful if you want to make a global change in a drawing, such as changing the width of all lines. If you want to specify that only certain objects, layers, or colors are selected you should use the **Selection Mask** command in the **Edit** menu.

This command in the **Edit** menu selects all objects except those on a hidden layer or excluded by the **Selection** mask. You can also double-click the **Selection** tool to select everything or use **Ctrl+A** to select all objects. If you choose **Select All** while using the **Text** tool, all characters in the current text area are selected.

Selection Mask

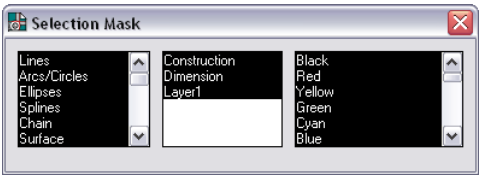
This command in the **Edit** menu allows you to limit selection by *object type*, *layer*, and *color*. You can only select objects that are highlighted in the dialog box.

The selection mask is useful when you have created a complex drawing and want to change particular groups of objects. You can use this feature to export some but not all geometry.

For example, if circles are not highlighted, when you choose **Select All** from the **Edit** menu, everything but the circles is selected. In this way, you can select such combinations as only blue splines or only red objects on a particular layer.

Using the selection mask

1. Choose **Selection Mask** from the **Edit** menu.
   The **Selection Mask** dialog box appears.



   The highlighted items respond to all selection methods and can be detected by the Drawing Assistant.
2. Click the items you want to be able to select so they are highlighted.

While the dialog box is visible, you can select, create, and edit geometry. You can move the dialog box if it covers geometry you want to work on. The selection mask remains in effect even after you close the dialog box until you select a tool.

When an item in the dialog box is not highlighted, the *Drawing Assistant* and all the tools cannot detect it, even though it is visible to you on the screen.

Deselecting

To deselect an object, click anywhere in the drawing area where there is no object, or click any tool in the tool palette.

You can deselect an object that was selected in a multiple selection

operation. While the objects are still selected, hold down the **Shift** key and click the objects you want deselected.

To deselect a list quickly, click one item in the list.

You can select a contiguous group by ***dragging*** or holding down the **Shift** key and clicking.

If you want to select or deselect non-contiguous items, hold down the **Ctrl** key and click the items. You select all items in a list by double-clicking one item in the list..

Selecting Points

Selecting points differs from selecting objects because points are not always visible.

Being able to select points is useful in two situations:

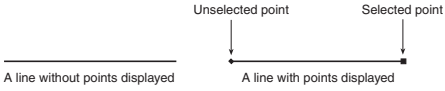
- stretching selected geometry (described later in this chapter)
- control point selection for transformations.

When you ***drag*** a selection marquee around objects, all geometry that is completely within the marquee is selected. If geometry is partially within the marquee, only the control points inside the marquee are selected, and the geometry is not selected.

Show Points

This command in the **Layout** menu toggles the display of the *control points* (endpoints, midpoints, center points, and knot points) for selected objects. When points are displayed, you can select a point by clicking it. If points are not displayed (but **Selectable Points** is set in the **Edit** menu), you can select a point by ***dragging*** a selection marquee around the location of the point.

To show points for an individual selection, use the **Edit Objects** command in the **Edit** menu.



Displaying points

1. Select the geometry.
2. Choose **Show/Hide Points** from the **Layout** menu to toggle the display of points *on* and *off*.

To turn off the point display once the points for an object are showing, you must select the object again and choose either **Show/Hide Points** from the **Layout** menu or the **Control Points** option in the **Edit Objects** dialog box.

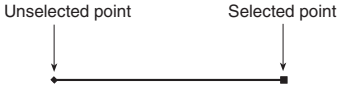
Showing and Hiding Points with Stroke

When you hold down the **Shift** and **Ctrl** key and click an object, the display of the object's points toggles on or off. If the points are hidden when you click the object, the points will be displayed.

Selecting invisible points

You can select a control point whether the points are visible or not; however, note that **Selectable Points** in the **Edit** menu must be on (displaying the ✓ symbol in the menu). To select points if points are not visible:

1. Click the **Selection** tool.
2. ***Drag*** a selection marquee around the location of the point.
   The selected point displays as a square.



Selecting visible points

To select points if they are visible:

1. Click the **Selection** tool.
2. Click the point.
   The selected point displays as a square.

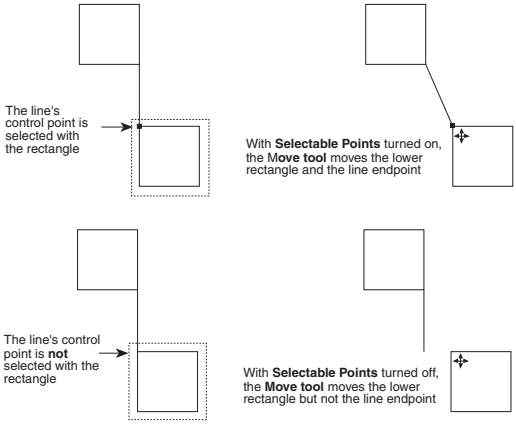
Selectable Points

This command in the **Edit** menu gives you the ability to select points that aren't displayed. When **Selectable Points** is not set, you cannot select points by ***dragging*** a marquee.

If points are displayed, simply click the point to select it. If points are not displayed, you can select a point by ***dragging*** a selection marquee around it.

The following example illustrates the use of **Selectable Points**.

If you select and ***drag*** a line, the line and the end-points move. If you select and ***drag*** only an endpoint of the line, the endpoint moves and the line length changes while the other endpoint of the line remains fixed.



CHAPTER

7

Editing Objects

Editing Tools

Moving Objects with Tools

Copying Objects with Tools

Sizing Objects with Tools

Editing Commands

Duplicating Objects

Changing the Characteristics of Objects

Grouping Objects

Arranging Geometry

Editing Objects

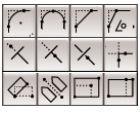
Once you create an object, you can select and alter it by choosing a command or selecting a tool to edit it. This section describes the common editing activities you can perform on a selected object.

The chapter also compares the different methods for performing the same action, such as the difference between ***dragging*** a selected item and using the **Move** tool.

All procedures in this chapter are described for single objects, they naturally are valid as well for several selected objects.

Editing Tools

These tools allow you to change an object physically, either altering the geometry, changing the size, or changing the location or orientation of the selected object. Three subpalettes on the tool palette provide these editing capabilities.



Fillet and Chamfer Tools
Trim Tools
Transformation Tools

Fillet and Chamfer	The *Fillet and Chamfer* tools round the corners of nonparallel lines or cut them at straight angles, respectively.
Trim	The *Trim* tools cut off or extend a line to the boundary limit you specify. You can also segment a line at an intersection, or create corners from intersecting lines.
Transformation	The *Transformation* tools *move*, *rotate*, *expand* or *shrink*, *stretch* and *mirror* objects.

Fillet and Chamfer Tools

These tools on the tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. The fillets and chamfers are automatically trimmed. If you do not want them trimmed, hold down the **Ctrl** key while you are selecting the objects to fillet or chamfer.

2-Entity Fillet Tool

This tool constructs an arc tangent to the two objects you click.

Before you can fillet *Smart walls* you have to ungroup them.

Adding a 2-Entity Fillet

1. Click the **2-Entity Fillet** tool.
2. Enter the arc radius you want in the status line. The default radius is **5 mm.**
3. Click the objects you want to fillet. You can also hold down the **Shift** key and click once inside the corner you want to fillet.

When filleting two parallel lines the fillet will be tangent to all objects only if the filleting radius is half the distance between the two parallel lines.

If you hold down the **Ctrl** key while you select the objects to fillet, the objects are not trimmed.

Fillets use the smallest arc between the selected geometry.

The status line allows you to specify the **R**adius of the fillet either before or after you create the fillet.

3-Entity Fillet Tool

This tool constructs a fillet tangent to the three objects you choose.

Adding a 3-Entity Fillet

1. Click the **3-Entity Fillet** tool.
2. Click the three objects you want to fillet.

When you fillet two parallel lines the filleting radius has to be half the distance between the two parallel lines.

If you hold down the **Ctrl** key while you select the objects to fillet, the objects are not trimmed.

There are no entries on the status line.

2-Entity Chamfer Tool

This tool creates a chamfer across a corner at the specified distance from the intersection of two lines. The default distance is **5** mm.

Adding a chamfer

1. Click the **2-Entity Chamfer** tool.
2. In the status line, enter the distance you want the chamfer from the corner. The default distance is **5** mm.
3. Click each line making up the corner you want chamfered. You can also hold down the **Shift** key and click once inside the corner you want to chamfer.

The lines are automatically trimmed or extended. If you hold down the **Ctrl** key while you select the objects to chamfer, the objects are not trimmed.

The status line allows you to specify the distance (**L**ength) from the chamfer to the intersection of the corner lines.

Angular Chamfer

This tool creates a chamfer at the specified angle and distance from the corner. The specified angle is the angle between the chamfer and the second line of the corner. The specified length is the distance between the corner and the intersection of the chamfer and the second line of the corner. The default distance is **0.5 mm** and the default angle is **45°**.

Adding a chamfer

1. Click the **Angular Chamfer** tool.
2. In the status line, enter the **length** you want the intersection of the chamfer and the second line of the corner from the corner. The default length is **0.5 mm**.
3. In the status line, enter the angle you want between the chamfer and the second side. The default angle is **45°**.
4. Click each line making up the corner you want chamfered. You can also hold down the **Shift** key and click once inside the corner you want to chamfer.

The lines are automatically trimmed or extended. If you hold down the **Ctrl** key while you select the objects to chamfer, the objects are not trimmed.

The status line allows you to set the distance (**L**ength) from the intersection as well as the angle.

Trim Tools

The *Trim tools* on the tool palette lengthen or shorten lines and curves. In general, you select the limiting object(s) before you select the tool. However, if you don't select everything you need, you can hold down the **Shift** key and click additional objects after the *Trim tool* is selected.

Simple Trim Tool

This tool shortens a line to the specified boundary. If you hold down the **Ctrl** key while using this tool, it becomes the **Relimit** tool.

Rule: Point to what you want to throw away.

Trimming objects

1. Select the object that limits the trim.
2. Select the **Trim** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the section of the object to be discarded.

There are no status line entries.

Relimit Tool

This tool lengthens or shortens a line to the specified boundary. If you hold down the **Ctrl** key while using this tool, it becomes the **Trim** tool.

Rule: Point to what you want to keep.

Relimiting objects

1. Select the object that limits the change.
2. Select the **Relimit** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the section of the object that you want to remain.

There are no status line entries.

Segment Tool

This tool divides a line or curve at intersections with other lines or curves.

Segmenting an object

1. Select the objects that limit the segmentation.
2. Select the **Segment** tool.
   If necessary, use **Shift**-Click to select more boundary objects.
3. Click the object to be segmented.

The selected object is segmented at the boundary objects. Even though you can not see the segmentation on the screen, you can select parts of the segmented line by clicking.

In order to see the segmentation on the screen, first select the boundaries and the object you want to segment and then **Show points** in the **Layout** menu. After the segmentation, the endpoints of the segmented parts are displayed.

If you hold down the **Ctrl** key while selecting the line to be segmented, the new segment appears in the characteristics of the *current pen* style.

There are no status line entries.

Corner Trim Tool

This tool creates a corner from the two objects you specify. Lines are extended or shortened to create the corner.

Creating a corner

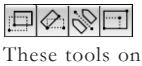
1. Click the **Corner Trim** tool.
2. Click each object. You can also press and hold the **Shift** key and click inside the about-to-be-created corner.

There are no status line entries.

Extending lines to a theoretical intersection

If you want to extend a line to its theoretical intersection with another line, first click the line to be extended, then hold down the **Ctrl** key and click the line that is not to be trimmed.

Transformation Tools



These tools on the tool palette *move*, *rotate*, *expand* or *shrink*, *stretch* and *mirror* objects. Select the object you want to transform before you select the *Transformation tool*.

You can copy at the same time you transform objects by holding down the **Ctrl** key while you specify the transformation.

When you press the **Shift** key you can select additional objects after you have selected a Transformation tool. As soon as you release the **Shift** key the transformation tool is active again.

Move Tool

This tool moves the selected objects to a new location. You can copy the selection by holding down the **Ctrl** key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

How to enlarge or shrink an object with the **Move** tool is described under *Sizing objects with Tools* in this chapter.

Moving objects

A positive or negative value entered in the status line determines the direction along the **X** or **Y** axis. A negative value moves the object to the left or down on the screen. A positive value to the right or up.

1. Select the object(s) to be moved.
2. Select the **Move** tool.
   If necessary, use **Shift**-Click to select more objects.
3. ***Drag*** the selected object to a new location, pressing to set a reference point and releasing to indicate the new location.

You can also click a reference point, then click a destination point to move the selected object to the new location. It is not necessary for the reference and destination points to be on the object you are moving.

The move is performed relative to the specified points.

The status line allows you to specify the **X**,**Y** distance that the selection is to be moved.

Rotate Tool

This tool rotates one or more objects around a specified point. You can copy the selection by holding down the **Ctrl** key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Rotating objects

1. Select the object(s) to be rotated.
2. Select the **Rotate** tool.
   If necessary, use **Shift**-Click to select more objects.
3. Specify the **center** of rotation.
4. ***Drag*** the objects or enter an **angle** in the status line to specify the rotation.

You can also click the *reference* point and then click the *destination* point. It is not necessary for the reference and destination points to be on the object you are moving. If they are not, the rotation is performed relative to the specified points.

The status line allows you to specify the **A**ngle of rotation.

Mirror Tool

This tool creates the mirror image of an object or objects on the opposite side of a reference line. You can copy the selection by holding down the **Ctrl** key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

1. Select the object(s) to be mirrored.
2. Select the **Mirror** tool.
   If necessary, use **Shift**-Click to select more objects.

If the mirror axis should be aligned in a precise angle, you have to create a construction line before.

3. Specify the *reference line* by clicking two locations or by ***dragging***. The *reference line* need not be parallel to the object.

The status line has no entries.

Expand/Shrink Tool

This tool enlarges or reduces objects while maintaining the proportions. You can copy the selection by holding down the **Ctrl** key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Using the beginning and ending reference point for the **Expand/Shrink** tool is particularly useful if you want to change the size of an object relative to the size of another object.

Expanding or shrinking objects

1. Select the object(s) to be expanded or shrunk.
2. Select the **Expand/Shrink** tool.
   If necessary, use **Shift**-Click to select more objects.
3. Click a point on the object that you want to remain stationary.
4. ***Drag*** the selected object to a new location, pressing to set a reference point and releasing to indicate the new location.

When you want to change the size of an object relative to the size of an other object you should click the *Reference and Destination point* instead of ***dragging*** the *Reference point*.

The status line allows you to enter the exact scale (**F**actor).

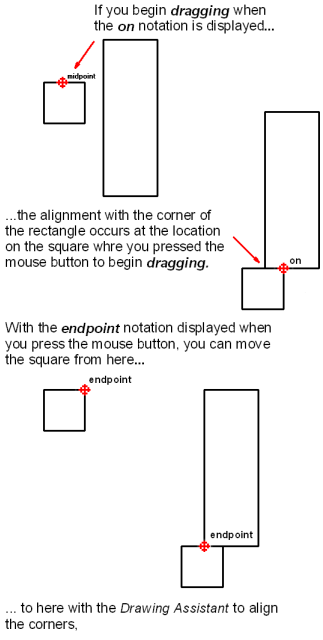
Moving Objects with Tools

When an object is selected, move the pointer near the object until the **Move** symbol appears and ***drag*** the object to a new location.

Using the Drawing Assistant for Moving

When the pointer becomes the **Move** symbol, you can ***drag*** the object around and see the *Drawing Assistant's* notations relative to the object's location when you pressed the mouse button.

In this way, if you move the pointer over a control point so the *Drawing Assistant* activates it, you can align the object you are moving with that point.

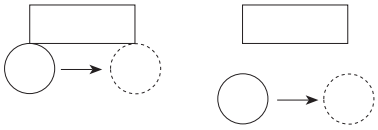


Selected Move versus Move Tool

You can move objects either with the **Selection** tool or with the **Move** function.

Moving with the Selection Tool

The move function of the **Selection** tool allows free movement of the selected object.

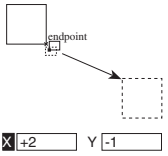


To move a circle from one location to another as shown above, use the **Selection** tool and the *Drawing Assistant* to align the 90° ***quadrant*** with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

Moving with the Move Function

The **Move** tool (on the **Transformation** subpalette) allows you to move the selection with reference to other geometry.

Move a **1 cm** square **2 cm** in the x direction and **1 cm** in the y direction as in the following graphic:



1. Select the square to be moved.
2. Select the **Move** tool.
3. With the **Move** tool, click one corner of the square when the ***endpoint*** notation is displayed.
4. Enter in the **X** entry box on the status line **+2** and in the **Y** entry box **-1**.
5. Press the **Enter** key.
   The square is moved, similar to the graphic above.

7 A positive or negative value entered in the status line determines the direction along the **X** or **Y** axis. A negative value moves the object to the left or down on the screen. A positive value to the right or up.

Copying Objects with Tools

You can copy selections with the **Copy** command as discussed earlier in this section or by holding down the **Ctrl** key while using the following tools:

- Selection tool
- Single Line tool
- Center-Point and Opposite-Point Circle tools
- Ellipse tools
- Polygon tools
- Text tool
- Transformation tools

Copying with the Selection Tool

You can hold down the **Ctrl** key and ***drag*** a copy of the selection to a new location.

Copying with the Selection Tool

1. Click the **Selection** tool.
2. Select the object or objects you want copied.
3. Hold down the **Ctrl** key.
4. ***Drag*** a copy of the selection to a new location.
5. Release the **Ctrl** key.

Copying with the Geometry Tools

You can make copies with the following drawing tools:

- Single Line tool
- Center-Point and Opposite-Point Circle tools
- Ellipse tools
- Polygon tools
- Text tool

Copying with the drawing tools

1. Construct the geometry you want to copy.
2. Hold down the **Ctrl** key.
3. Click a new location. The click determines the location of the first point specified during the construction of the original geometry (the center of a center-point circle, for example).
4. Release the **Ctrl** key.

Copying with the Transformation Tools

You can hold down the **Ctrl** key while using a **Transformation** tool to make a copy of the selected geometry.

Copying with the Transformation Tools

1. Select the object you want to copy and transform.
2. Select one of the four **Transformation** tools.
3. Hold down the **Ctrl** key.
4. Perform the transformation according to the directions in the message line.
5. Release the **Ctrl** key.

   A copy is transformed and the original remains unchanged.

Copy Option versus Copy Command

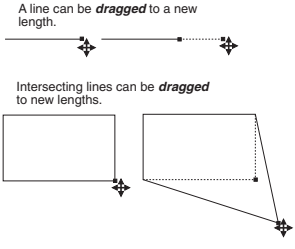
Usually, using the copy option of a geometry tool is faster than using the **Copy** and **Paste** commands in the **Edit** menu. The **Copy** command is very useful for copying to a different document or application.

Sizing Objects with Tools

Normally, you size an object with the **Selection** tool or the **Expand/Shrink** tool. In some cases it is useful to use the move tool for sizing objects.

Sizing an object with the Selection Tool

You can stretch objects by selecting a point and ***dragging*** it to a new location.



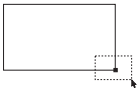
1. Be certain that **Selectable Points** is set in the **Edit** menu.
2. Click the **Selection** tool.
3. ***Drag*** a selection marquee around the *control points* that represent the area you want to stretch.
4. ***Drag*** the points to a new location.

Sizing an object with the Move Tool

You can also size an object with the **Move** tool. Using the **Move** tool allows you to specify the distance the selected point(s) should be moved by specific values along the **x** and **y** direction in the status line.

To move the corner of the rectangle in the next graphic with the **Move** tool you proceed as follows:

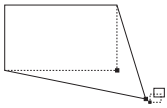
1. Be certain that **Selectable Points** is set in the **Edit** menu.
2. Click the **Selection** tool.
3. ***Drag*** a selection marquee around the lower right corner of the rectangle.



4. Click the **Move** tool.
5. Enter **+1** in the **X** entry box and **-1** in the **y** entry box in the status line.
6. Press the **Enter** key.

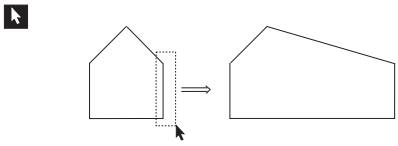
   The corner of the rectangle is moved the distance you have specified in the status line.

Use the **Move** tool for sizing objects only once since the next values you enter in the status line refer not to the new position of the moved point(s), but **always** to the original one. That means if you enter (in our example) **-1** in the **X** entry box and **+1** in the **Y** entry box, the point is not moved back to its original position, but in the opposite direction referring to its original position.

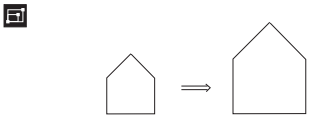


Selected Sizing versus Expand/Shrink Tool

Dragging a control point of a selected object changes the size of the object, but it also distorts the object, changing the proportion between height and width.



The **Expand/Shrink** tool on the **Transformation** subpalette enables you to resize geometry while maintaining its proportions.



Use the **Expand/Shrink** tool to enlarge or shrink a shape proportionally

In addition, you can specify proportions by clicking points on other geometry. For example, you can resize an object to fit within another object by clicking the boundary into which the resized object must fit.

Editing Commands

The difference between the **Cut** and **Copy** commands is that cutting a selection removes it from the drawing area while a copied selection remains in the drawing area.

Four commands in the **Edit** menu allow you to change objects without changing the physical geometry. You can copy or move objects within the same documentation. You can also copy or move objects to other documents, even to other applications.

The **Cut**, **Copy**, and **Paste** commands do not alter the object's attributes -the layer and pen style (*color*, *weight*, and *pattern*). If you want to change the attributes of an object, use the **Edit Objects** command in the **Edit** menu.

Cut Ctrl+X

This command in the **Edit** menu removes the selected objects and places them on the *Clipboard*. Each selection you cut or copy to the *Clipboard* replaces the previous *Clipboard* contents.

Cutting objects

You can also erase selections with the **Delete** or **Backspace** key or by choosing **Delete** from the **Edit** menu. Selections erased in this way do not go on the *Clipboard* and cannot be pasted. You can restore them, however, by using **Undo**.

1. Select the object to be cut.
2. Choose **Cut** from the **Edit** menu.

Once you cut a selection, you can paste it. You can use **Cut** and **Paste** to move geometry or text around the document, from one sheet to another, or from one document to another. You can even paste the cut selection into a document in a different application.

Moving geometry with the Cut command

1. Select the objects you want to move.
2. Choose **Cut** from the **Edit** menu.
3. Display the location where you want the selection to appear in the drawing area, scrolling if necessary.
4. Choose **Paste** from the **Edit** menu.

If you want to move an object from one layer to another, use the **Edit Objects** command in the **Edit** menu.

The object appears in the center of the drawing area on the original layer on which it was created or onto the work layer if it's from a different application. The object is selected so you can move it, if you want.

Copy Ctrl+C

This command in the **Edit** menu places a copy of the selection onto the *Clipboard* without deleting the original selection. You can paste the copy elsewhere in the current document or into a different document. You can even paste the copied selection into a document created with a different application.

You can also create a copy of an object or text box with the **Selection** tool. First select the object, then hold down the **Ctrl** key and ***drag*** a copy of the object to a new location.

Copying objects

1. Select the objects to be copied.
2. Choose **Copy** from the **Edit** menu.
   The selection goes on the *Clipboard*, and it remains in the current document.

You can copy an object using any of the **Transformation** tools by holding down the **Ctrl** key while using the tool.

Paste Ctrl+V

This command in the **Edit** menu pastes a copy of the Clipboard contents onto the center of the drawing area. The Clipboard contents are not changed when you use the **Paste** command.

You can also paste the selection into another document or into a document created with another application.

If you want to create an even distribution of geometry, you can use **Linear Duplicate** or **Polar Duplicate** in the **Edit** menu.

If the object was cut or copied from a ***DraftBoard*** document, when it is pasted, it goes on the layer it was on when it was cut or copied. A pasted selection retains its original attributes. Objects cut or copied from non-***DraftBoard*** documents are pasted onto the work layer.

The Clipboard

The **Cut** and **Copy** commands both place a copy of the selected object on the *Clipboard*. The *Clipboard* is a buffer —a temporary storage place that holds the last cut or copied selection. The contents of the *Clipboard* are objects, not bitmaps.

Erasing Geometry

You can use several methods to erase objects:

- You can select the object(s), then press the **Backspace** key.
- You can retrieve what you deleted with the **Undo** command.
- You can undo the last eight actions.
- You can select the object(s), then choose the **Delete** command from the **Edit** menu. You can retrieve what you deleted with the **Undo** command.
- You can select the object(s), then choose **Cut** from the **Edit** menu. You can retrieve what you cut by using the **Paste** command, ***as long as you have not cut or copied anything else***.

Delete

This command in the **Edit** menu deletes a selection without placing it onto the *Clipboard*. You can **Undo** this deletion, but you cannot paste what is deleted. In addition to this command, you can use the **Delete** or **Backspace** key to delete a selection.

Retrieving with Undo and Redo

Once an object is deleted or cut, you can retrieve it with the **Undo** command in the **Edit** menu. **Undo** can reverse each of the last eight actions. You can use **Ctrl+Z** to issue the **Undo** command from the keyboard.

The number of **Undo's** and **Redo's** is limited to **32**.

Undo Ctrl+Z

This command in the **Edit** menu reverses the last action. You can **Undo** and **Redo** any of the last eight actions. For example, if you delete an object, you can choose **Undo** to restore it. You can then use **Redo** to return to the deleted version.

You can undo actions that create and edit geometry and text but not actions that do not change the contents of the drawing, such as resizing the window or quitting.

If you are using a tool which involves a multistep process, such as constructing connected lines or creating a 3-point fillet, choosing **Undo** reverts to the beginning of the process. Some other activities can be stopped by pressing the **Escape** key.

Redo Shift+Ctrl+Z

This command in the **Edit** menu reinstates the last action reversed by **Undo**. You can **Undo** and **Redo** any of the last eight editing actions.

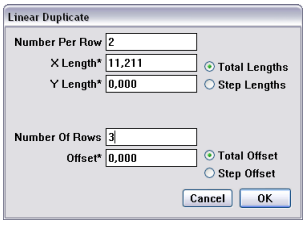
Duplicating Objects

DraftBoard allows you to create multiple copies of an object and arrange them in an array along a straight line or in a circular pattern.

Linear Duplicate

This command in the **Edit** menu allows you to create an array of copies along a straight line.

An asterisk (*) appearing beside an entry field indicates that you can fill in the box by ***dragging*** the mouse in the drawing area to indicate the length.



The **Linear Duplicate** dialog box contains the following settings:

Number Per Row — The total number of objects you want in each row. Be sure to include the selected object itself in this total count.

X Length/Y Length — The **X Length** value determines the horizontal length of a row. The **Y Length** value determines the vertical rise of the row. The **X Length** and **Y Length** can be set by entering values, or by clicking the mouse in the drawing area.

Total Lengths — The **X** and **Y Length** is the distance from the selected object to the last copy.

Step Lengths — The **X** and **Y Length** is the distance from the selected object to the first copy.

Number of Rows — The total **Number of Rows** that you want.

Total Offset — The offset is the perpendicular distance between the row containing the selected object and the **last** row.

Step Offset — The offset is the perpendicular distance between the row containing the selected object and the **next** row.

Offset — The perpendicular distance setting for the **Total Offset** or **Step Offset**. You can specify the Offset by entering a value or by clicking the mouse in the drawing area.

Using Linear Duplicate

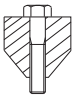
If you begin a duplication and discover that it is not what you want, you can stop the process by pressing the **Escape** key, and then choosing **Undo** to remove the copies from your document.

1. Select the object to be duplicated.
2. Choose **Linear Duplicate** from the **Edit** menu.
3. Enter the **number** of objects **per row**.
4. Specify **Total** or **Step** Lengths.
5. Click the **X Length*** box.
6. Move the pointer to the drawing area and drag to indicate the length of the row of copies.
   Both the **X** and **Y** values appear in the entry boxes.
7. If you want more than one row, enter the number of rows, and specify the type of offset.
8. Enter the offset, if any. You can use a negative number.
9. Click **OK**.

Dragging to indicate the offset is particularly useful if you want the offset to equal the distance between two objects.

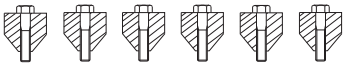
Example

Suppose you want to illustrate six machine screws. You could draw geometry, then use **Linear Duplicate** to copy the geometry in an array.



1. Create and select the geometry you want to duplicate.

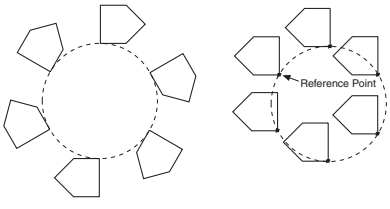
2. Choose **Linear Duplicate** from the **Edit** menu.
3. Enter **6** in the **Number Per Row** box.
4. Click the **Step Length** button.
5. Click in the **X Length** box.
6. Move the pointer into the drawing area and drag across the widest point of the step.
7. Click at the end of the entry in the **X Length** box.
8. Type ***2** to multiply the width of the geometry by two.
9. Click **OK**.



The geometry is duplicated in a linear array.

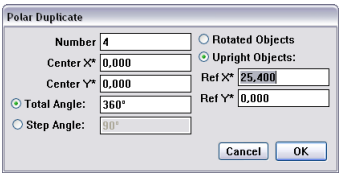
Polar Duplicate

This command in the **Edit** menu allows you to copy and rotate the selected geometry. When you duplicate an object in a circular array, you can specify the number of duplications, the center of the array and whether the objects are copied in a rotated orientation or an upright orientation.



Rotated Objects Upright Objects

When you select the **Polar Duplicate** command in the **Edit** menu the following dialog box is displayed:



An asterisk (*) appearing beside an entry field indicates that you can fill in the box by ***dragging*** the mouse in the drawing area to indicate the length.

The **Polar Duplicate** dialog box contains the following settings:

Number — The total number of objects you want. Be sure to include the selected object itself in this total count.

Center X/ Center Y — The **Center X** and **Y** values determine the center of the circular array.

The **X/ Y** values can be also dragged on the drawing area

Total Angle — The total number of degrees between the center of the selected object and the center of the last copy.

If the copies should be arranged semicircular the *Total Angle* must be **180°**.

Step Angle — The number of degrees between the center of the selected object and the center of the first copy.

If you want to place an object three times in a semicircle, the *Number* must be **3** and the *Step Angle* **90°**.

Rotated Objects — Copies are rotated relative to the angle between the copies. Each single arbitrary point of all copies have the same distance to the center of the circular array. That means that any point of the object can be the *reference* point.

If the option **Rotated Objects** is selected, in principle any point on the object becomes a *Reference Point.*

Upright Objects — Copies are upright with respect to the selected object. This option requires a reference point.

Ref X/Ref Y — The *reference* point for the **Upright Objects** option.

Ref X and **Ref Y** determine an imaginary point duplicated around the center, as specified. The selected objects are reproduced in the same position relative to each imaginary point that is duplicated. With the option **Rotated Objects** the reference point has a constant distance to the center of the circular array.

The **Ref X-** and **Ref Y** coordinates can be specified by entering the values in the respective fields or clicking a point on the drawing area.

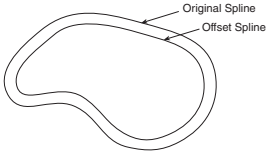
Using Polar Duplicate

If you begin a duplication and discover that it is not what you want, you can stop the process by pressing the **Escape** key, and then choosing **Undo** to remove the copies from your document.

1. Select the object to be duplicated.
2. Choose **Polar Duplicate** from the **Edit** menu.
3. Enter the **number of objects** in the circular array.
4. Click the **Center X*** box.
5. Move the pointer to the drawing area and click to indicate the center for the array of copies.
   The values for **Center X** and **Center Y** appear in the entry boxes.
6. Specify **Upright** or **Rotated** objects by clicking a button.
7. If you specified **Upright**, click a location for the reference point in the drawing area.
8. If you do not want the copies in a complete circle, click the **Total** or **Step Angle** button.
9. Enter a value for the specified angle type.
10. Click **OK**.

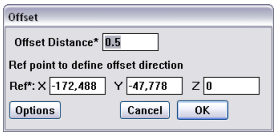
Offset

This command in the **Edit** menu allows you to create offsets from *lines*, *arcs*, *circles*, *ellipses* and *splines*.



7 The asterisk indicates that you may use the cursor to input an offset distance by ***dragging*** between any two points on the drawing area. The cursor must be in the **Offset Distance** field while ***dragging***.

When you select the **Offset** command in the **Edit** menu the following dialog box is displayed:



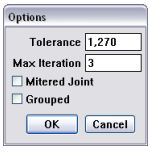
The **Offset** dialog box contains the following settings:

The offset **Direction** is also filled in the respective entry fields when you ***drag*** the offset distance since the vector you ***drag*** determines the **Offset Direction** automatically.

Offset Distance The offset distance is the distance away from the original geometry that the new geometry will be created. For multiple offsets additional values may be entered manually into the **Offset Distance** field, separated by semicolons.

Ref. The **X** and **Y** values can be filled in manually or indicated by a mouse click on the drawing area offset from the object (the **Z** value must be **0**). The point clicked determines which side of the object the offset should be performed.

Clicking the **Options button** displays the following dialog box:



The **Options** dialog box contains the following settings:

Tolerance For *Spline* and *Ellipse* objects the offset distance is approximated by a spline. By setting a **Tolerance** for this type of object you can determine how accurate the offset is relative to its theoretical position.

Max Iteration ***DraftBoard*** will iterate when calculating the offset of *Spline* and *Ellipse* objects to improve the approximation the number of times indicated in the **Max Iteration** field. The default value is **3**. Setting a high value will decrease performance.

Mitered Joint If you mark this option the offset object will have square external corners instead of filleted corners. This option is by default not selected. The radius of

the filleted corners is automatically set to the offset distance. For instance; a box that is offset by **.25** will have a fillet at each corner with a radius of **.25**.

Grouped If this option is marked the new offset geometry will be grouped when created. This option is by default not selected to produce ungrouped geometry.

The geometry will be only grouped, if the **Grouped** option was checked.

Creating an Offset

1. Select the object you want to offset.
2. Select the **Offset** command in the **Edit** menu.
3. Click into the **Offset Distance** field.
4. Input an offset distance by ***dragging*** between any two points on the drawing area.
5. Click into one of the **Ref** entry fields.
6. Input an offset direction by ***dragging*** between any two points on the drawing area.
7. Click the **Options** button.
   Specify in the **Options** dialog box a **Tolerance**, if needed and whether or not the object will be offset with mitered corners. The new geometry will be created as a group and will leave the original geometry selected.

Changing the Characteristics of Objects

You can edit selected objects by either changing individual characteristics, such as layer or line style, or by changing the specifications in the **Edit Objects** dialog box. The **Edit Objects** dialog box also provides information about the selected object in addition to allowing you to make changes.

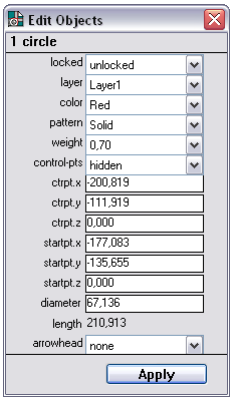
You can change a single characteristic faster by simply selecting the object and changing the characteristic. But if you are changing several characteristics or any measurements or positions, you should use the **Edit Objects** command.

Edit Objects Ctrl+I

This command in the **Edit** menu allows you to edit selected objects by changing individual characteristics, such as *length*, *layer*, or *pen style*, or other specifications. Changes made through this dialog box can be reversed with the **Undo** and **Redo** commands.

Editing objects

1. Select the object to be edited.
2. Choose **Edit Objects** from the **Edit** menu.
   The dialog box appears.



Many boxes have pop-up menus for selection. Press the down arrow and then ***drag*** to the selection you want.

3. Change the information in the entry boxes.
   Double-click the entry box and type a new entry (do **not** press **Enter**).
4. Make any other changes you want.
5. Click the **Apply** button.
6. Double click the **Close** box in the upper left corner to close the **Edit Objects** dialog box.

In the **Edit Objects** dialog box you can use mathematical operators. A list of all allowed operators is in the *Appendix* of this manual.

You can use **Undo** and **Redo** to reverse changes made through this dialog box.

The specifications shown in this box depend on the type of object selected, and include at least the following:

- Number (or type) of objects
- Lock status
- Current layer
- Current color
- Current pattern
- Current weight
- Absolute coordinates for the starting point and ending point of the object

You can use different units in each entry box like inch ('), feet ('), feet/inches (x'y"), centimeter (**cm**) millimeter (**mm**) or Meter (**m**). You are also allowed to mix English and metric units as long as they are labelled correctly like 10" + 25.4 cm.

The measurements reflect the settings of the **Units** option in the **Preferences** submenu. If you want to prevent changes to an object, you can specify **locked** in this dialog box, or you can select the object and choose **Lock** from the **Arrange** menu.

If you are doing a lot of editing, you may want to leave the **Edit Objects** dialog box displayed. In that way you can select the object, make the changes in the dialog box, click **Apply**, and then go on to the next object.

If more than one object is selected when you choose **Edit Objects**, only the common information is displayed. Entries are blank when the information isn't common. For example, if you select two concentric circles, the center will be displayed in the **Edit Objects** box but the diameter box will be blank.

If you select several objects and then choose **Edit Objects**, all of the objects will reflect the changes you make in the dialog box. For example, if you select all dimensions and change the text entry in the Edit Objects dialog box to **2**, all the dimensions will display a **2**.

Moving an object to a different layer

1. Select the object(s) to be moved.
2. Choose **Edit Objects** from the **Edit** menu.
3. Click the arrow for the **layer** entry box.
4. Click the layer from the list.
5. Click **Apply**.

Arranging Geometry

The display order set with the **Bring to Front** and **Send to Back** commands is valid as well when selecting objects.

The **Arrange** menu contains some commands that enable you to subdivide geometry, group geometry so it behaves like a unit and lock geometry to prevent changes . In addition, this menu contains a command to refresh the screen after you have changed geometry.

This menu contains also commands to sort objects on the screen. When using these commands objects in the foreground will cover objects in the background. This feature allows to manipulate the default drawing order of ***DraftBoard***, that puts always that object into the foreground that was drawn last.

Bring to Front

This command can be used as well when placing text in front of a filled surface.

This command brings a selected object into the foreground and will cover all underlying objects. When this command is selected for another object, the object brought into the foreground before, will go one position backwards. This allows to sort objects specifically on the screen. If several sorted objects are grouped, the sorting order will be kept, even if the grouped object is send to front again.

Send to Back

With multiple hierarchical grouped objects that functions only partly.

This command sends a selected object into the background and will be covered by all objects lying in the foreground. When this command is selected for another object, the object brought into the background before, will go one position forwards.

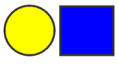
This allows to sort objects specifically on the screen. If several sorted objects are grouped, the sorting order will be kept, even if the grouped object is send to back again.

Bringing objects to Front/Back (Example)

1. Draw a circle using the line color **Black** and a line weight of **1,5 mm**.

2. Fill the circle with the color **Yellow**.
3. Select the yellow fill area (without the black perimeter).
4. Select the **Send to Back** command in the **Arrange** menu.
   The yellow fill area is send to back and covered by the black perimeter circle.

5. Select the circle and the fill area and choose the **Group** command in the **Arrange** menu.
   The yellow fill area and the circle perimeter are grouped as sorted.
6. Draw a rectangle with the color **Black** and a line weight of **1,5 mm**.
7. Fill the rectangle with the color **Blue**.
8. Select the blue fill area (without the black rectangle line).
9. Select the **Send to Back** command in the **Arrange** menu.
   The blue fill area is send to back and covered by the black perimeter lines.



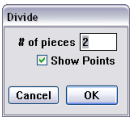
10. Select the **Group** command in the **Arrange** menu.
    The blue fill area and the perimeter lines are grouped as sorted.
11. Select the blue rectangle and move it partly over the yellow circle.
12. Deselect the blue rectangle by clicking anywhere onto the drawing area.
    The blue rectangle in the foreground covers the yellow circle.

13. Select the yellow circle.
14. Select the **Bring to Front** command in the **Arrange** menu.
    The yellow circle is brought to front and covers the blue rectangle.

Divide

This command in the **Arrange** menu subdivides the selected geometry into the specified number of equal parts.

1. Select the object to be divided.
2. Choose **Divide** from the **Arrange** menu.
   The following dialog box appears.



3. Enter the number of equal parts you want to divide the selection into.
4. Click **OK**.

You can see the divisions when you display the points by clicking **Show Points** in this dialog box or choosing **Show Points** from the **Layout** menu.

Group Ctrl+Y

This command in the **Arrange** menu combines selected objects to function as a single object.

If you move the group, all components move together. If you change the size of a group, the individual objects change proportionally.

Grouping objects

1. Select the objects to be grouped.
2. Select **Group** from the **Arrange** menu.

Once you group geometry, you can't edit the individual objects within the group unless you ungroup them.

To create temporary groups, you can ***drag*** a selection marquee around several objects to treat them as a single unit while they are selected.

You can also combine groups. For hierarchical groupings, ***DraftBoard*** ungroups each group in the order in which they were combined.

Changing the members of a group without adding a hierarchical level

When you want to add new geometry to an existing group, and you select the group and the new geometry and choose the Group command, you end up with a group within a group. If you want to make a single group, follow these directions:

1. Select the group.
2. Choose **Ungroup** from the **Arrange** menu.
   The geometry is ungrouped with the individual objects selected.
3. Hold down the **Shift** key and select the geometry you want to add to the group.
4. Choose **Group** from the **Arrange** menu.

You can follow a similar procedure to remove members of a group.

Ungroup

This command in the **Arrange** menu separates selected grouped objects into their individual components.

Ungrouping Objects

1. Select the group.
2. Choose **Ungroup** from the **Arrange** menu.
   The individual objects in a group are selected.

Lock

This command in the **Arrange** menu prohibits editing or movement of selected geometry.

Preventing accidental changes to one or more objects

1. Select the objects to be locked.
2. Choose **Lock** from the **Arrange** menu.

Although you cannot move or change a locked object, you can copy, group, and select it.

Preventing changes to an entire document

1. Choose **Select All** from the **Edit** menu.
2. Choose **Lock** from the **Arrange** menu.

Selected objects can also be locked or unlocked by clicking the **Locked** box in the **Edit Objects** dialog box.

Unlock

This command in the **Arrange** menu removes the lock in the selection. You can change and move the unlocked objects.

Unlocking Objects

1. Select the locked object(s).
2. Choose **Unlock** from the **Arrange** menu.

Selected objects can also be locked and unlocked by clicking the **Locked** box in the **Edit Objects** dialog box.

CHAPTER

8

Text, Fills and Crosshatches

Text

Crosshatching and Solid Fills

Text, Fills and Hatches

This chapter describes the features you can use to add detail to your drawing: *text*, *crosshatching* and *solid fills*.

Text

When you want to annotate your drawing, you'll use the **Text** tool to create a text box for entering characters from the keyboard. You can create, import, and edit text on screen. You can use the **Text** menu to set the characteristics, such as *font*, *text size*, *style*, *alignment*, *line spacing*, and *indentation*.

Text Tool

This tool on the tool palette allows you to create and edit text on screen. The text you enter has the characteristics set in the **Text** menu.

Creating text

1. Select the **Text** tool from the tool palette.
2. Create a text box by ***dragging*** or by clicking two locations to place the opposite corners of the text area.

   The text box which appears is as tall as a single line of text and as wide as you indicate. The text cursor appears in the box, ready for your typing.
3. Enter the desired text from the keyboard.

   Pressing the **Delete** key erases characters to the left of the cursor.

   The text you enter automatically *word wraps* to the next line when it reaches the right side of the text box. If you resize the text box, the words rewrap automatically.

Importing a text file

You can import an **ASCII** file that contains text you want to use in your drawing. For example, you can place specifications you've written with a word processor on your drawing without retyping it. The entire file is imported, so if you want to use only part of the document, create a new file that contains only the information you want entered on the drawing.

An **ASCII** text file is usually created by word processing software and saved as *plain text*. It usually contains only standard keyboard characters (*letters*, *numbers*, *punctuation*, and *spaces*) without the formatting typically found in word processing documents.

1. Select the **Text** tool from the tool palette.
2. Create a text box by ***dragging*** or clicking twice to place the opposite corners of the text area.
3. Choose **Import** from the **File** menu.

   A dialog box appears.
4. Select the text file you want to import.

   The **Import** dialog box appears and the **Text** file type is specified.
5. Click **OK**.

   The text file appears in the text box you created.

Editing text

If you want to change characteristics for the entire text entry, use the **Selection** tool to select the text block.

1. Select the **Text** tool from the tool palette.
2. Move the pointer directly over the text to be edited.

   The pointer turns into an **I-beam** text cursor when it is over text. (This only occurs when the **Text** tool is active.)
3. Use the text cursor to select text.

   You can ***drag*** either direction to select characters, and double-click on an individual word to select the word.

   You may also simply click to place the text cursor within the existing text if you want to add more to it.
4. Modify the selected text by choosing the appropriate commands from the **Text** menu.

Changing the characteristics of a text block

If you want to change the characteristics of a word or portion of a text block, use the **Text** tool to select what you want to change.

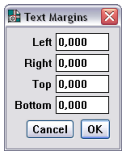
You can change the *font*, *text size*, *style*, *spacing*, and *indentation* of a text block.

1. Use the **Selection** tool to select the text block to be changed.
2. Select the new characteristic from the **Text** menu.
   The selection shows the new characteristic.

Modifying text margins

1. Choose the **Text Margins** command from the **Preferences** submenu in the **Layout** menu.
   The following dialog box is displayed.

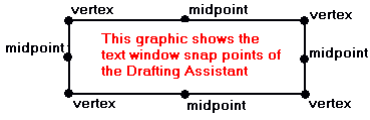
Text margins of selected can be also edited with the **Edit Objects** command in the **Layout** menu.



2. Enter values for the *Left*, *Right*, *Top* and *Bottom* text margins.
   This values are valid for all future text placed with the **Text** tool from the *Function palette*. You can save these margins permanently using the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

Moving Text

Each text object has **nine** snap points for the *Drawing Assistant*. These snap points will help you to move and/or align text with the **Selection** tool.



1. Choose the **Selection** tool from the Tool palette.

2. Select the text object you want to move.
3. Move the cursor over one of the nine text snap points. The cursor becomes a ***4-way Move*** *symbol*.

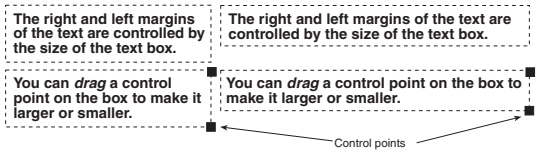
4. Place the ***4-way Move*** *symbol* at the desired snap point using the *Drawing Assistant* notations as a guide.
5. ***Drag*** the text object to the new location.

Changing the size of the text box

1. Click the **Selection** tool.
2. ***Drag*** a selection marquee around the right side of the text box.
3. ***Drag*** the control points to the left or right.
   The area changes size and the text rewraps automatically.

Selectable Points in the **Edit** menu must be active for selecting the control points of the text box.

When you want to set the size of the text box accurately you select the text box with the **Selection** tool and choose the **Edit objects** command in the **File** menu. In the **Width** and **Height** entry boxes you can enter values for the box size.



Moving Text and Files into the DraftBoard drawing area (Drag&Drop)

Text can be moved directly from all programs that support **OLE** into the ***DraftBoard*** *drawing area*.

DraftBoard *Files* (documents with the file extension **.vlm**) will be directly opened, when moved from the ***Windows****-Explorer* or from the ***Windows*** *Desktop* into the ***DraftBoard*** *drawing area*.

Dragging text into the DraftBoard Drawing area

1. Create text using a text editor (such as ***MS****-Word)* that supports **OLE**.

2. Select the text and ***drag*** it with the left mouse button pressed into the ***DraftBoard*** *drawing area*.
3. Release the left mouse button.

   The text is placed on the ***DraftBoard*** *drawing area* at the location where the mouse button was released. The text is displayed using the current ***DraftBoard*** text font and all paragraph formats will be maintained.

Dragging DraftBoard Files into the DraftBoard Drawing area

1. Select a ***DraftBoard*** *File* in the ***Windows*** *Explorer* or on the ***Windows*** *Desktop*.
2. ***Drag*** the ***DraftBoard*** *File* with the left mouse button pressed into the ***DraftBoard*** *drawing area*.
3. Release the left mouse button.

   The ***DraftBoard*** *drawing* is automatically opened in ***DraftBoard*** .

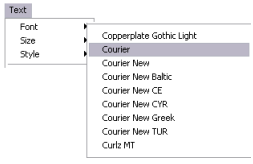
Text Menu

The **Text** menu contains commands for changing the *text font*, *size*, and *style*, as well as the *justification*, *spacing*, and *indentation*.

A *checkmark* in front of a menu item shows the current attribute.

Font

This command in the **Text** menu changes the *font* for selected text or future text entries.



Text can be rotated with the **Rotate** function like any other object in a drawing including text using *Postscript* and *True Type* fonts.

The fonts listed in the menu are those installed on your computer and include ***DraftBoard's*** plotter fonts.

Use ***Plotter*** *fonts* whenever you are creating text for a drawing that you intend to send to a plotter, since plotting *Postscript* or *TrueType* fonts take much more time to print or may be substituted.

The font you choose stays in effect for the current document until you choose another font.

Special characters and accents are available as described in the *Appendix* of this manual.

Extended font selection

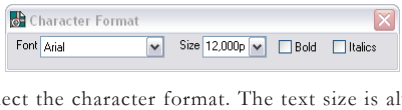
The **Font** submenu can display up to *20 fonts*. If you have installed more than 20 fonts select **More** in the **Font** submenu of the **Text** menu to get access to all available fonts.

Choosing **More** from the **Font** submenu brings up a dialog box allowing you to specify the *font*, *text size* and *text style*. The text size is measured as point size. If you want to specify the size in the units you have set for your drawing, you have to type the exact unit such as **"** for inch or **mm** for millimeter after the value.

In addition to the plotter fonts shipping with ***DraftBoard*** you can use all plotter fonts that are compatible with the file format **.shx** of AutoCAD©. When you copy these files into the **Fonts** folder of the ***DraftBoard*** folder, the new fonts are displayed in the **Fonts** submenu after relaunching ***DraftBoard***.

1. Choose **More** from the **Font** submenu in the **Text** menu.

   The **Character Format** dialog box displays:



2. Select the character format. The text size is always measured as point size (**pt**). If you want to specify the size in another unit like in **Millimeter**, include **mm** with your numeric entry.

All selections made in this dialog box take effect immediately for all selected text or text boxes.

All menu items in the **Text** menu affect only text created with the **Text** tool. Text attributes for dimensions are set in the *dimension editor* under **Preferences** in the **Dimension** menu.

However when you type in a value (only possible in the **Size** box) the change becomes effective only until you do another selection or continue to work on the drawing area.

You can leave this dialog box open to assign other character formats. If you want to close this dialog box, you have to double-click the **Close box** in the upper left corner of the dialog box.

The character format you specify stays in effect until you choose another character format.

Printing TrueType Fonts on Pen Plotters

When you want to print ***TrueType*** *Fonts* on pen plotters you must use a ***WinLINE Plus*** *Plotter Driver*. Look for more information about ***WinLINE*** *Plus* in the Internet under ***www.winline.com***.

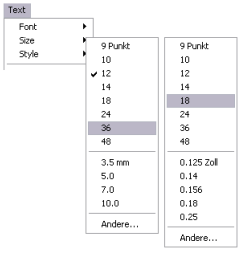
Font Sizes and ANSI Standards

A typical font specified for **7,0** mm may produce a character that is only **3,5** mm. The variation depends on the proportional spacing of the font. If you want a font set to exact specifications, use a **Plotter** font, which conforms to **ANSI** standards.

Size

This command in the **Text** menu sets the font size for the selected text or for future text entries in the current document. The size can be specified as either points or units, as defined in the **Units** dialog box of the **Preferences** submenu in the **Layout** menu.

If you have selected *American* units under **Units** in the **Preferences** submenu of the **Layout** menu the *text size* is displayed in **Point** and **Inches**, when you have selected *metric* units it is displayed in **Point** and **Millimeters**.



8

Specifying a nonstandard text size

Choosing **Other** from the **Size** submenu brings up a dialog box allowing you to specify the exact text size you desire in whatever units you've set for your drawing. You can specify point size by entering **pt** after the value in the entry box; for example, if you enter **11 pt** the font size will be 11 point.

All items in this dialog box affect only text created with the **Text** tool. Text attributes for dimensions are set in the *Dimension Editor* under **Preferences** in the **Dimension** menu.

1. Choose **Other** from the **Size** submenu in the **Text** menu.

2. Enter the size you want. The size is measured in the current units as set in the **Preferences** submenu. If you want to specify a point size, include **pt** with your numeric entry.
3. Click **OK**.

   The size you specify stays in effect until you choose another size.

Text Size and Scaling

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

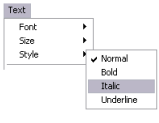
A detailed description about *text size* and *scaling* can be found in ***Chapter 9*** *Basic Drafting*.

When you scale a drawing using the **Drawing Size** command without the options *keep text size* and *keep dimension text size* after you have added *text* and *dimensions*, you have to assign the correct text size again in the **Text** and **Dimension** menus.

This is not necessary when you add *text* and *dimensions* **after** scaling or scale the drawing with the options *keep text size* and *keep dimension text size* active.

Style

This command in the **Text** menu sets the style (such as *bold* or *underline*) for selected text or future text entries.



The style you specify stays in effect until you choose another style. A check (✓) indicates the current style. You can change the default setting by saving the changes in the preferences file ***Prefs.vlm*** with the **Save Preferences** command in the **Preference** submenu of the **Layout** menu.

Text Alignment

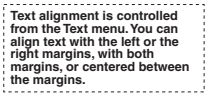
The four commands in the second section of the **Text** menu align your text within the text box. The text can be aligned on the *left*, the *right*, *centered* in the *middle*, or it can be fully *justified* so that it is aligned with both left and right margins of the text box.

If the **Text** tool is active, both the selected text and future text are aligned as specified, but existing text is not aligned. If you choose this command when a tool other than the **Text** tool is active, only future text entries are aligned. If a text entry is selected when you choose this command, the selection and future text entries are aligned as specified.

Align Left

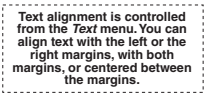
This command in the **Text** menu aligns the selected and future text at the left margin of the text area.

When you want to set the *text offset* from the box boundary you have to select the text box with the **Selection** tool and choose the **Edit objects** command in the **File** menu. You can enter values for the *text offset* in the related boxes.



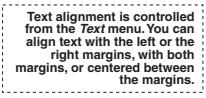
Align Middle

This command in the **Text** menu centers the selected and future text in the text area.



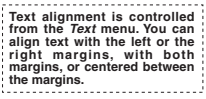
Align Right

This command in the **Text** menu aligns the selected and future text at the right margin of the text area.



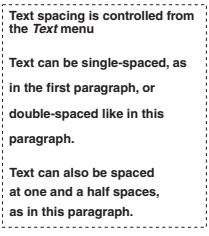
Justified

This command in the **Text** menu aligns both the left and right margins of the selected and future text in the text area.



Text spacing

The **Text** menu has commands for *single spacing*, *double spacing*, or using *a space and a half* between the lines of your text.



You can change the default setting by saving changes in the **Preferences** file.

The check (✓) in the **Text** menu indicates the current line spacing.

Single Space

This command in the **Text** menu sets the spacing of selected and future lines of text so each line occupies one space in the text area.

1-1/2 Space

This command in the **Text** menu changes the spacing of selected and future text so each line occupies one and a half spaces in the text area.

Double Space

This command in the **Text** menu sets the spacing of selected and future text so each line occupies two spaces in the text area.

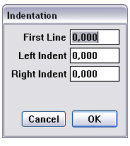
Indentation

If you wish to indent the text on the right or left side of the text box, or if you wish to have the first line indented, choose **Indentation** from the **Text** menu.

Indentation

This command in the **Text** menu sets the indentation of paragraphs for the selected text area as defined by the units set in the **Preferences** submenu.

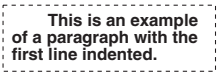
1. Choose **Indentation** from the **Text** menu.
   The dialog box appears.



2. Specify the indentation you want. You can use point values if you include **pt** in your entry.
3. Click **OK**.

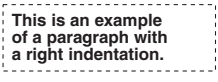
You can specify the number of units for any or all the choices:

First Line Sets the number of units for the indentation of the first line of each paragraph.



Left Indent Sets the number of units for the indentation of the left margin of each paragraph.

Right Indent Sets the number of units for the indentation of the right margin of each paragraph.



Crosshatching and Solid fills

DraftBoard can *crosshatch* and *fill* any enclosed area in your drawing and then automatically update the crosshatching when you change the dimensions of the enclosed area. You can specify a closed area that has a hole or other cutout in it, and ***DraftBoard*** accurately excludes the hole area from the crosshatching and filling.

You can choose the **Fill** command from the keyboard with **Ctrl+B** and the **Hatch** command from the keyboard with **Ctrl+H**.

In the **Pen** menu you will find two fill commands: **Fill** and **Solid Fill** and two crosshatch commands: **Crosshatch** and **Hatch**. **Fill** creates a solid fill for a selection with the current pen color and **Hatch** crosshatches the selection with the current pattern. If you want to change the patterns, choose **Crosshatch** and specify the pattern you want to use. If you want to change the default hatch pattern, make changes to the *preferences file* as described in *The Basics* chapter.

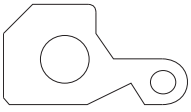
Hatch and Fill boundaries

The best way to select a hatch boundary is with the **Tracer** tool.

Before you can *hatch* or *fill* an area you must select the *hatch* or *fill boundaries*. You can select the boundaries with the **Selection** tool or the **Tracer**.

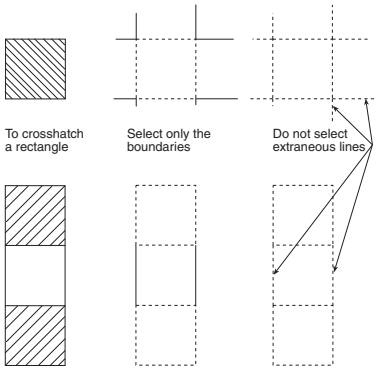
Specifying the Fill or Crosshatch Area

The selected objects must completely enclose the area you want to crosshatch or fill. For example, to crosshatch the object below, you would select all of the lines making up the part, including the holes.



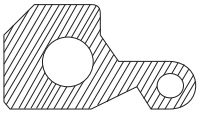
The exterior lines define the boundary for the crosshatching or solid fill. The two holes tell ***DraftBoard*** that those enclosed areas within the original area should not be crosshatched or filled.

*It is important that you do not select extra objects when you specify the crosshatch or fill area because **DraftBoard** does not know how to treat the extra objects.*



Hatching and Filling

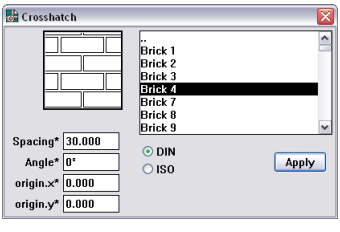
Once you have selected the lines and holes, you can choose the **Hatch**, **Fill**, **Crosshatch** or the **Fill** command from the **Pen** menu and specify the pattern you want to use.



Crosshatch

Using this command you can select a hatch pattern. You can change the default hatch pattern by saving the changes with the **Save Preferences** command in the **Layout** menu under **Preferences**.

Choosing the **Crosshatch** command the following dialog box is displayed.



The **Crosshatch** dialog box allows the following settings:

Patterns In the **Crosshatch** list you select a pattern. Depending which option is checked **DIN** or **ISO** the **Crosshatch** list may contain subdirectories with additional hatch patterns, that you can open with a mouse double-click.

How to add and define your own hatch pattern is described at the end of *Chapter 4* (Preferences) and in the *Appendix* of this manual.

Spacing In this entry field you can set the *spacing* of the hatch pattern. If you don't add a unit that unit is valid that you specified under **Units** in the **Preferences** submenu of the **Layout** menu.

Angle In this entry field you can enter a value for the *angle* of the hatch pattern.

Origin X/Y* In these entry fields you specify values for the **X-** and **Y-Coordinate** of the *Hatch* ***Origin***. You can define these values as well by ***dragging*** the mouse on the ***DraftBoard*** *drawing area*. You just must click in one of the two entry fields and then on the intended point on the drawing area. As soon you release the mouse but-

ton, the corresponding values are automatically filled into the entry boxes.

Pattern box The **Pattern** box shows the revised pattern exactly as it will be when the specified object is crosshatched. If you have scaled a drawing, double click in the pattern box to adjust the displayed pattern.

Apply When you click **Apply** all selected and all future objects will be hatched with the settings you specified.

Adding crosshatching

To crosshatch a figure that is not a closed figure, you must add lines or segment lines to make a closed figure. In this way, you can crosshatch individual sections of the geometry. See *Crosshatching* in the *Advanced Features* section of the Tutorial for a step-by-step example.

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Crosshatch** from the **Pen** menu.
3. Select the pattern you want from the crosshatch list.
   The pattern you select appears in the **Pattern** display area as it will appear on the drawing.
4. Change the spacing and angle, if you want.
   As you make changes to the angle or spacing, the pattern box shows the revised pattern exactly as it will be when the specified object is crosshatched.
5. Click **Apply**.

Specifying a Hatch Origin

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Crosshatch** from the **Pen** menu.
3. Select the pattern you want from the crosshatch list (for example **Bricks 8** in the folder **Extended**).
4. Change the spacing and angle, if you want.
   As you make changes to the angle or spacing, the pattern box shows the revised pattern exactly as it will be when the specified object is crosshatched.
4. Specify in the two entry fields **Origin X*** and **Origin Y*** a value for the new origin of the hatch pattern. Alternatively you can click first in one of the two fields and then in the lower left corner of the hatch boundary on ***DraftBoard*** *drawing area* for the new origin of the selected hatch pattern. The values are automatically filled in into the two **Origin** fields.
5. Click **Apply**.
   The origin of the hatch pattern is moved accordingly.

Editing an existing crosshatch pattern

You can edit hatch patterns with the **Edit Objects** command in the **Edit** menu.

1. Select the existing crosshatching by clicking it with the **Selection** tool.
2. Choose **Crosshatch** from the **Pen** menu.
3. Make the changes you want in the **Crosshatch** dialog box.
4. Click **Apply**.
   The crosshatching changes as you specified.

Hatch Ctrl+H

This command is located in the **Pen** menu and crosshatches the selected objects using the current crosshatch pattern. If you want to change the crosshatch pattern, choose **Crosshatch** and make your selection from the dialog box.

Crosshatching with the current pattern

1. Select the object you want to crosshatch.
2. Choose **Hatch** from the **Pen** menu.
   The selected area is crosshatched with the current pattern.

Solid Fill

This command in the **Pen** menu fills the selection with the *current pen color* and the *fill tolerances* you specified.

Creating a Solid Fill

The best way to select a fill boundary is with the **Tracer** tool.

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.
2. Choose **Solid Fill** from the **Pen** menu.
   The **Solid Fill** dialog box displays:

3. Set a **Tolerance** for the fill.
   The tolerance controls how smooth the linearization is for fills in *circles*, *arcs*, *ellipses* and closed *splines*. Increasing the value of **Tolerance** makes the fill smoother. The default value is **0.01**. (A value of **0.001** will produce smoother edges than **.01**. This is also effected by Drawing Scale.)

A solid filled object will only cover permanently its background if it is brought to front with the **Bring to front** command in the **Arrange** menu.

4. Click **Apply**.
   The selected object is filled with the current pen color.

Changing the color of an existing solid fill

To change the tolerance of an existing fill, first select the fill by clicking on it. Choose **Solid Fill** from the **Pen** menu. Change the **Tolerance** and press **Apply**. You can also change the **Tolerance in the** Edit Objects dialog box.

1. Select the existing solid fill by clicking it with the **Selection** tool.
2. Choose a new color from the **Color** pop-up menu on the **Pen** menu.
   The **Fill** color changes as you specified and will be visible as soon you deselect the solid fill.

Fill Ctrl+B

This command is located in the **Pen** menu and fills the selected objects using the *current pen color* and *tolerance*. If you want to change the *tolerance*, choose **Solid Fill** and make your selection from the dialog box.

Solid filling with the current pen color and tolerance

1. Select the object you want to fill.
2. Choose **Fill** from the **Pen** menu.
   The selected area is filled with the current pattern and tolerance.

Adding Crosshatch Patterns

DraftBoard offers a wide variety of *DIN* and *ANSI hatch patterns* with editable pattern *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box.

A graphical compilation of all crosshatch patterns you find in the *Appendix* of this manual.

Adding Hatch Patterns

All hatch patterns that are compatible with the hatch format **.pat** of AutoCAD© can be used in ***DraftBoard***.

You only must copy hatch files with the *.pat file extension into the ***DraftBoard*** folder **DraftBoard\Scripts\VSG\Hatches**. The file name is then automatically displayed in the ***DraftBoard*** **Crosshatch** dialog box. If the file contains more than one hatch pattern, a folder with this file name is created, that contains all hatchpatterns stored in the ***.pat** file.

Specifying Crosshatch Patterns

You specify your own hatch patterns by creating and binding in *hatch description files* into ***DraftBoard***.

Hatch patterns are defined by *hatch pattern lines,* that are running through a defined origin and are endless repeated at a defined offset distance.

Hatch pattern lines are infinitely long. Each *Hatch pattern line* may contain *single pattern elements* to create dashed or dotted pattern lines. A *Crosshatch pattern* has as well an infinite size and a hatched geometry shows only part of this pattern like a detail window.

Creating Hatch description files

To bind in your own hatch pattern into ***DraftBoard*** you must save the *Hatch description files* as ***ASCII*** *files* into the **DraftBoard** folder

..\Scripts\ VSG\ Hatches

A *Hatch description file* can contain the description of one or several crosshatch patterns.

The file name of a *Hatch description file* is displayed in the **Crosshatch** dialog box of ***DraftBoard***, if it is copied in the folder mentioned above. The file extension of this ***ASCII*** *file* must be always

***.pat**

Without this file extension ***DraftBoard*** is not able to identify this file and cannot display the crosshatch patterns described in this file.

To create a *Hatch description files* you can use any *Text Editor* such as ***NotePad*** for *Windows*.

Pattern Definitions

A *Hatch description file* contains several sections. The following example explains the file structure describing a simple pattern at a specific angle:

```
;;
;;  DraftBoard
;;  Definition Angle Hatch Pattern
;;
*ANGLE, Angle steel
0, 0,0, 0,.275, .2,-.075
90, 0,0, 0,.275, .2,-.07
```

Each file starts with a *Comment line*, that is indicated by a semicolon (**;**) before each comment.

All lines starting with a semicolon are ignored by ***DraftBoard*** when interpreting the pattern description.

The Comment line is followed by the *Header line* that starts with an asterisk (*****) followed by the pattern *name* and a pattern *description*. Name and Description must be separated by a *comma* (**,**)or a *space* ( ).

The **Crosshatch** dialog box of ***DraftBoard*** shows only the *name* not the *description*.

The name must be unique in the file. If it isn't ***DraftBoard*** uses the first pattern it finds by that name.

The length of the *Header line* is limited to a maximum of 80 characters for *Name*, *Comma* and *Description*.

The header line is followed by the *Hatch data*. Every line of data uses the same format:

angle, x-origin, y-origin, x-offset, y-offset, [, dash1, ...]

Angle defines the angle that this line of hatch pattern data is displayed at. **O** means that the hatch line is drawn horizontally. **90** means that the line is drawn vertically.

Comma separates the numbers.

x-origin The first line of the hatch pattern passes through this x-coordinate.

y-origin The first line of the hatch pattern passes through this y-coordinate.

x-offset defines the distance between line segments; the gap distance. You only use this parameter to specify the offset for vertical or diagonal lines (To specify the distance between dashes, use the **dash1** parameter. In most hatch patterns, the *x-offset* has a value of **0.0**; however, even though this parameter is rarely used, it is not optional.

y-offset defines the vertical distance between repeating lines; this parameter is used by every hatch pattern.

Dash1 You can use one or more **dash1** parameters to define dashes in the hatch pattern line. The **dash1** parameter is optional and not required for solid lines. A positive number, such as **.25** defines the length of a dash. A dot drawn by

the hatch pattern is defined by the value **0**. But to be sure that dots will be printed by any type of printer it is recommended to define for dots a very short **dash1** parameters close to **0**.

Tips on creating Pattern Codes

Each line of code applies to a single pattern segment. The two lines of data in our example represent a hatch pattern with two lines.

There is no limit to the number of data lines for hatch pattern definitions. But complex patterns can take a long time to draw on the screen.

You cannot specify a weight or width for hatch pattern lines. The work around is to define two or more very closely spaced lines.

A pattern line consists of straight lines and dots. You cannot specify arcs, circles and other round elements in a hatch pattern file. To simulate circular elements, use a series of very short dashes.

To make it easier to read your hatch pattern code use tabs and spaces to format the code into nice columns like this:

```
Angle x,y-origin  x,y-offset   dash codes
37.5, 0,0,        1.123,1.567  1,-0.5,2,-0.5
```

The hatch description file

Extended.pat

shipping can be modified for the creation of your own patterns.

You find this file in the ***DraftBoard*** folder

..\Scripts\ VSG\ Hatches

CHAPTER

9

Dimensions

Associative Dimensions

Dimension Tools

Dimension Settings

Dimensions

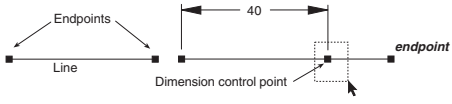
The **Dimension** menu contains commands for displaying the palette of dimensioning tools and for all dimension settings.

Associative Dimensions

DraftBoard's geometric dimensions are associative—when you make a change to the geometry, the dimension changes also.

This associativity is a tremendous time saver because dimensions automatically update whenever you make a change in the geometry. You can even change the units from *English* to *Metric* (one setting in the **Preferences** submenu in the **Layout** menu) and every dimension on your drawing will reflect the change.

When you extend a line by selecting the ***endpoint*** of the line and ***dragging*** it to a new position the dimension changes also, because the dimension has a *control point* at the same position of the line ***endpoint***. So when you select the ***endpoint*** of the line you also select the dimension *control point*.



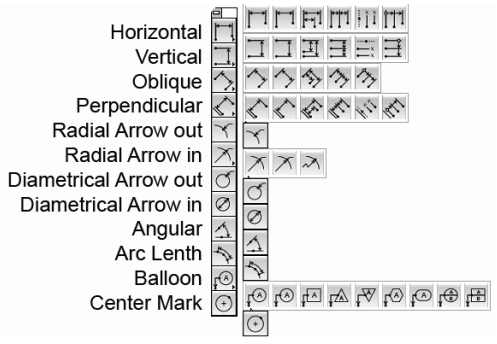
Dimensions are associative relative to the points they measure. If you change the length of a line using **Edit Objects**, the dimension will not update because the point was not changed. To correct the dimension, select the dimension's ***vertex*** point and ***drag*** it to the new ***endpoint*** of the line.

Using the Dimension Tools

DraftBoard's dimensioning tools are on a separate palette which you can ***drag*** around the drawing area. You select dimension tools in the same way you select tools from the tool palette. The first four tools have subpalettes which also work like the subpalettes of the tool palette.

Show Palette

This command in the **Dimension** menu displays the dimension palette on screen.



The dimensioning tools can be used to measure either an object or the area between objects. Once you have dimensioned an object (or space), the dimensions update when you make changes to the geometry.

The first four tools on the dimension palette have subpalettes for base line and chain style dimensions.

Some of the dimensioning tools, such as the *horizontal* and *vertical* tools, require that two points be selected; others, such as the *Radial* and *Diametral*, require only one.

Saving the Status of the Dimension Palette

The **Save Palette** command in the **Preferences** submenu of the **Layout** menu allows to save the status (*visible* or *invisible*) of the dimension palette including its *position* on the screen as default setting for any relaunch of ***DraftBoard***. Status and position are saved to the **DraftBoard.ini** file in the **Palette** section. When the palette is not visible after a restart, it could be that is was saved outside the screen area. In that case you enter into the **DraftBoard.ini** file in the **Palette** section the Pixel coordinates **20, 20** and start ***DraftBoard*** again.

Dimensioning objects and space

1. If necessary, display the **dimension** palette.
2. Select the appropriate dimension tool.

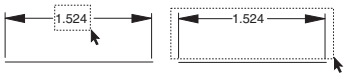
   The dimension pointer is a smart pointer with a *hot spot*—the dot shows which side of the object to select first. If you select as indicated by the position of the dot, the text appears above or to the right of the leader. If you select in the opposite order, the text appears below or to the left of the leader.
3. Click the points you want to measure.

   All settings for dimensions can be set in the *dimension editor* under **Preferences** in the **Dimension** menu.
4. Move the dimension to a new location if its placement is not where you want it.

Repositioning the dimension

When the dimension text appears, it is selected so that you can move it to a new location. Move the pointer to the dimension text. When the pointer changes to the **Move** symbol, ***drag*** the dimension to its new location.

If you want to move a dimension later, use the **Selection** tool. Click the dimension once, then ***drag*** it to the new location. You can also ***drag*** to select the dimension text or to select the entire dimension or several dimensions at once.



Using the dimension status boxes

The status boxes shown below appear when a dimensioning tool is selected.

Text The **#** symbol in the *text box* indicates that the dimension is the actual value of the object's measurement. If you delete this symbol, the dimension will no longer be associative (that is, it won't update if you change the geometry it measures).

If you are going to use *Parametrics* (only available in ***DraftBoard Unlimited***), you must delete the **#** symbol and enter values, algebraic expressions, or variables in the text box.

Some of the dimensioning tools, such as the **Radial** tools, add a letter in the text status box which appears in the dimension itself. You can add text before or after the **#** symbol. You can enter parentheses to enclose the resulting dimension in parentheses (**#**), or enter **# 2 Plcs** to display the dimension followed by the text **2 Plcs**. When you include the **#** symbol, ***DraftBoard*** displays the measurement of the geometry as well as the text you add.

See as well *Tolerances* in this chapter.

Tolerance The **Upper** and **Lower** boxes appear if you have specified a tolerance format in the *dimension editor* (under **Preferences** in the **Dimension** menu). You enter the limits for the tolerance in these boxes.

Placing dimensions automatically on a separate layer

1. Choose **Preferences** in the **Dimension** menu.
2. Select in the **Layer** *pop-up* of the *dimension editor* a layer that all dimensions should be placed on.

For the ***DIN Standard*** the **Dimension** layer is the default layer.

3. Close the *dimension editor* with a double-click on the **Control** menu.
4. Dimension as usual.

   Subsequent dimensions automatically go on the dimension layer rather than the work layer.

Adding tolerance and limits

1. Choose the desired style in the *dimension editor* (under **Preferences** in the **Dimension** menu).

2. Dimension as usual.
3. Enter values for the *upper* and *lower* tolerance limits in the status line and press **Enter**. It is also allowed to delete the default values and keep these fields empty or replace the values by text entries. The dimension will display always the content of the status line.

Using fixed dimensions

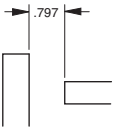
If you delete the **#** symbol in the text box (on the status line for a newly-created dimension or in the **Edit Objects** dialog box) and replace it with other text, the entry in the text box is fixed—it does not update if you change the units or the size of the geometry.

Horizontal Dimension Tools

These tools measure horizontal spaces or the distance between linear objects.

Horizontal Dimension Tool

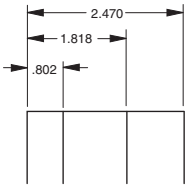
This tool dimensions an object or space horizontally. Click the left point first, then click the right point. (Click the points in the opposite order if you want the dimension to display below the objects.)



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

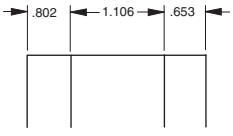
Horizontal Base Line Dimension Tool

This tool dimensions objects or spaces horizontally from a base point.



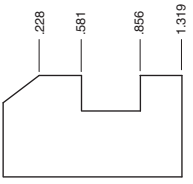
Horizontal Chain Dimension Tool

This tool dimensions objects or spaces from end-to-end, horizontally.



Horizontal Ordinate Dimension Tool

This tool dimensions objects or spaces horizontally from a base point.

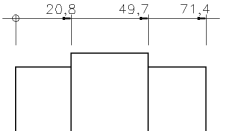


***Remember**: to display the dimension number of the base point you have to click it twice: at the **first** and the **last** dimension points.*

If you want to display the dimension number for the base point you have to click the base point again after you have clicked all other dimension points.

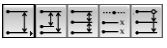
Horizontal Base Chain Dimension Tool

This tool creates a *horizontal* base chain dimension, displaying the rising distance between objects and spaces to a base point.



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Vertical Dimension Tools



These tools measure vertical space or the distance between vertical objects.

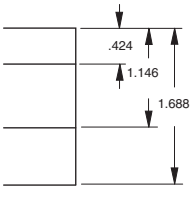
Vertical Dimension Tool

This tool dimensions an object or space vertically. Click the top point first, then click the bottom. (Click the points in the opposite order if you want the dimension to display to the left of the objects.)

When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

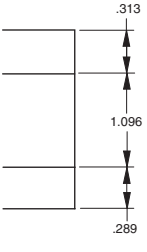
Vertical Base Line Dimension Tool

This tool dimensions objects or space vertically from a base point.



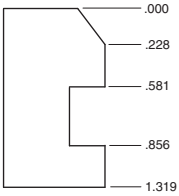
Vertical Chain Dimension Tool

This tool dimensions objects or space from end to end, vertically.



Vertical Ordinate Dimension Tool

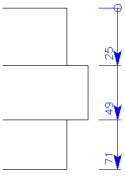
This tool dimensions objects or space vertically from a base point.



If you want to display the dimension number for the base point you have to click the base point **again** after you have clicked all other dimension points.

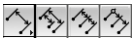
Vertical Base Chain Dimension Tool

This tool creates a *vertical* base chain dimension, displaying the rising distance between objects and spaces to a base point.



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

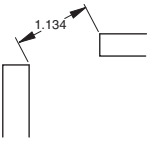
Oblique Dimension Tools



These tools measure space or objects, obliquely from point to point.

Oblique Dimension Tool

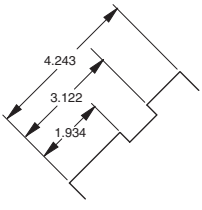
This tool dimensions an object or space from point to point obliquely. Click the left point first, then click the right. (Click the points in the opposite order if you want the dimension to display below the objects.)



When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

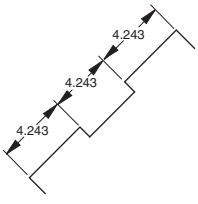
Oblique Base Line Dimension Tool

This tool dimensions objects and spaces point to point obliquely from a base point. ***Note****: The results shown reflect colinear dimension points.*



Oblique Chain Dimension Tool

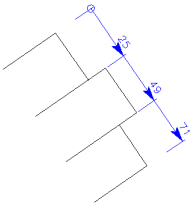
This tool dimensions objects and spaces oblique point-to-point, end-to-end.



Note: The results shown reflect colinear dimension points. Lines are *colinear* if they have a common invisible *carrier line*. ***DraftBoard*** can recognize colinear lines only if they touch at least in one end-point.

Oblique Base Chain Dimension Tool

This tool creates a *oblique* base chain dimension, displaying the rising distance between objects and spaces to a base point.

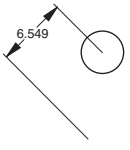


Perpendicular Dimension Tools

These tools measure an object or space perpendicular to another line. If most of your dimensions start from a line edge, you can use this tool to generate both horizontal and vertical dimensions.

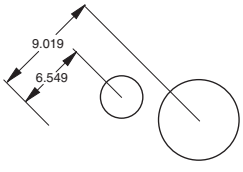
Perpendicular Dimension Tool

This tool dimensions an object or space perpendicular to a line. Click the base line first, then click the object or location.



Perpendicular Base Line Dimension Tool

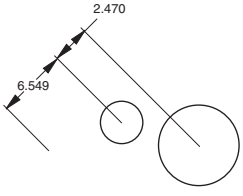
This tool dimensions between a point or object perpendicular to an existing base line.



Note: Be certain that you click on the base line, not on the endpoint of the base line. When the dimension text appears, you can **drag** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

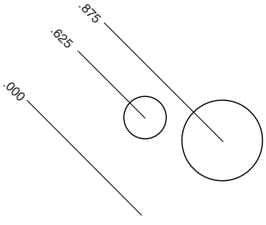
Perpendicular Chain Tool

This tool dimensions between points or objects perpendicular to an existing baseline.



Perpendicular Ordinate Dimension Tool

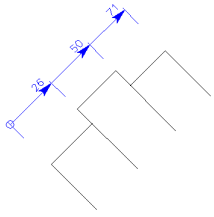
This tool dimensions between a point or object perpendicular to an existing base line.



If you want to display the dimension number for the base point you have to click the base point **again** after you have clicked all other dimension points.

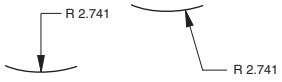
Perpendicular Base Chain Dimension Tool

This tool creates a *oblique* base chain dimension, displaying the rising distance between objects and spaces to a base point.



Radial Arrow Out Dimension Tool

This tool measures the radius of a circle, arc, or fillet with the arrow on the side of the arc indicated when the arc is picked. To dimension the radius of a circle, arc, or fillet click near it. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside.

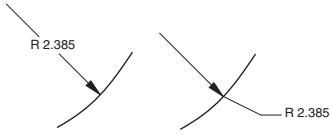


When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

You can move a dimension arrow to a new location by ***dragging*** a selection marquee around the *control point* at the top of the *dimension arrow* and then ***drag*** it to a new location.

Radial Arrow In Dimension Tool

This tool measures the radius of a circle, arc, or fillet with the arrow inside the geometry. To dimension the radius of a circle, arc, or fillet, click near it. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside. Either way the arrow line starts from the arc center.

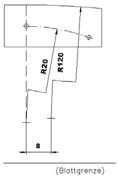


When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Extended Radial Arrow In Dimension Tool

This tool measures the radius of a *circle*, *arc*, or *fillet* with the arrow inside the geometry. To dimension the radius of a *circle*, *arc* or *fillet*, click near it (the *Drawing Assistant* must show ***on***). If you click outside it, the dimension text appears outside, and if you click inside, it appears inside. Either way the arrow line starts from the arc center.

In contrast to the two other radial dimensions the *Extended Radial dimension* allows to draw the dimension line of a large radius shortened and angled. Both, the part of the dimension line with the dimension arrow and the angled part of the dimension line, point to the geometrical center.



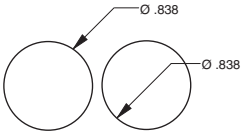
When the dimension appears the leader line is placed at the nearest ***1°*** *increment* from the location you clicked.

The leader line is always drawn to the geometrical center. To shorten and angle the leader line, you must first ***drag*** the dimension text to a new location and select then the end of the dimension line at the geometrical center and ***drag*** it away from the center point.

The dimension line is shortened and angled in a z-Form, where both legs point to the geometrical center.

Diametral Single Arrowhead Dimension Tool

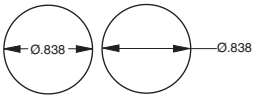
This tool measures the diameter of a circle. To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension text appears outside, and if you click inside, it appears inside.



When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Diametral Double Arrowhead Dimension Tool

This tool measures the diameter of a circle. To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension appears outside, and if you click inside, it appears inside.

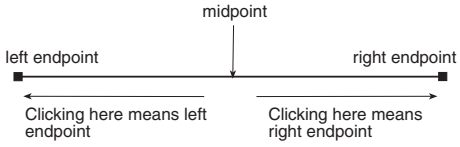


When the dimension appears the leader line is placed at the nearest ***15°*** *increment* from the location you clicked. You can ***drag*** the text to a new location.

Angular Dimension Tool

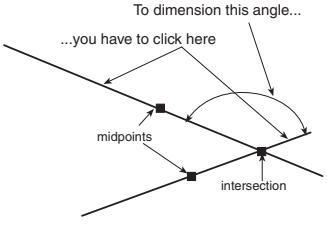
This tool measures the angle between two lines. To dimension an angle, click each line near the endpoint from which you want the angle measured. The inside angle is measured between the sides with the extension lines extending from the endpoints nearest the location you clicked on the line. The smaller angle is used.

The angle between the endpoints of the lines is measured so you do not have to click these endpoints, just near them. ***DraftBoard*** interprets
all mouse clicks left of the line midpoint as the left endpoint and all mouse clicks right of the line midpoint as the right endpoint.



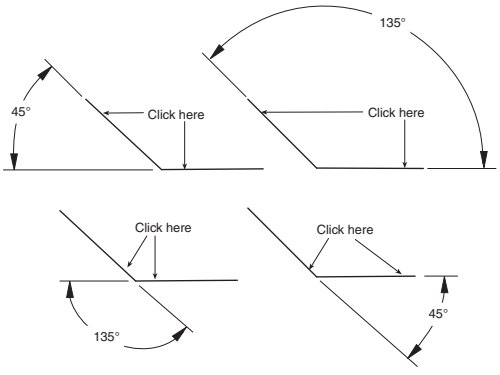
If you dimension intersecting lines don't mix up the intersection with the midpoint of the lines.

Angular dimensions will display all angles properly if you keep the angles larger than **3** degrees. For dimensions smaller than that, you should build the dimensions manually using lines and text objects.



The Angular **Dimension** tool will not dimension angles over 180°.

If the lines don't meet, the angle is measured from the invisible extension of the lines. The extension appears as part of the dimension.



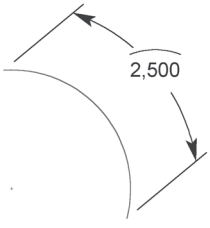
When the dimension text appears, you can ***drag*** it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Arc Length Dimension Tool

This tool measures the length of an arc. To use it, you have to move the cursor to the arc to be measured and click on the following points:

- one end of the arc,
- the middle of the arc and
- the opposite end of the arc (as if you were creating a three-point arc).

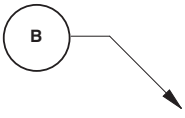
The **Arc Length Dimension** tool does not put witness lines perpendicular to the arc for angles less than **180** degrees.



Like other dimensions you can manipulate the text by changing its size or font using the *Dimension Editor*. You can move the text by ***dragging*** it to the desired position.

Balloon Symbol

This tool attaches a *Balloon symbol* to geometry, placed in the location you click.



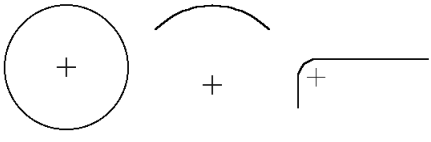
All *Balloon Symbols* use as line style **Balloon**. Therefore you should use this line style as well when creating *Balloon Symbols* manually .

You can enter text and the diameter of the balloon in the Status Line.

Center Mark

This tool marks the ***center*** of a *circle*, *arc* or *fillet*.

To mark the ***center*** of a *circle*, *arc* or *fillet*, click near it (the *Drawing Assistant* must show ***on***).



Dimension Settings

Scaling a drawing using the **Drawing Size** command affects always the *dimension appearance* regardless of the specified *dimension settings*.

Dimension Appearance

When scaling a drawing all *existing dimensions* are scaled as well. Only dimensions placed after scaling a drawing are displayed at their specified text size and are not adjusted to the current *view scale*.

Dimensions are always displayed correctly at their true size, when:

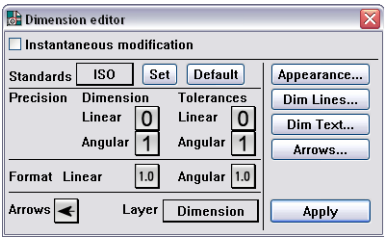
- the option *keep dimension text size* in the **Drawing Size** dialog box is checked while scaling
- a drawing is dimensioned **after** scaling
- or the original text size is set **after** scaling.

A detailed description about Scaling you find in ***Chapter 9*** *Basic Drafting* of this manual.

Dimension Editor

All attributes for dimensions created with the dimension tools are set with the *Dimension Editor*.

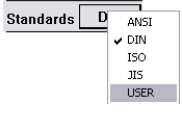
Choosing the **Preferences** command in the **Dimension** menu displays the *Dimension Editor*:



The *Dimension Editor* allows the following settings:

Standards

The **Standards** *pop-up* menu allows you to select one of the following dimension standards:



When you select a new standard the *dimension editor* displays the related settings.

All standards can be modified. The **Set** button saves all modifications made in the *Dimension Editor* to the current standard. These changes are valid until the end of the current ***DraftBoard*** session or until you modify the settings again. You can save the selected *standard* permanently using the **Save Preferences** command from the **Preferences** submenu in the **Layout** menu.

You can save the selected *standard* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Setting a dimension standard

1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. In the **Standards** *pop-up* menu ***drag*** to the standard you want to modify.
3. Modify the desired parameters in the *Dimension Editor*.
4. Click the **Set** button.
   All settings are valid until the end of the current ***DraftBoard*** session or the settings are modified again.
5. Click the **Apply** button if you want to assign the new settings to selected dimensions.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.
6. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.

All modifications made are only valid for the current standard. When you want to edit only some selected dimensions you must select the current standard again after you clicked the **Apply** button.

Restoring a dimension standard

1. Choose **Preferences** in the **Dimension** menu.
   The *dimension editor* is displayed.
2. In the **Standards** *pop-up* menu ***drag*** to the standard you want to restore to its default settings.

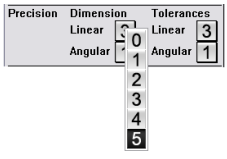
3. Click the **Default** button.
   All settings of the selected standard are restored to its default values.
4. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.

Standards and modifications

All modifications are assigned to the current standard. When you want to edit only some selected dimensions you have to select the standard again after you have clicked the **Apply** button to restore its original settings.

Precision

The **Precision** *pop-up* menus in the *Dimension Editor* allows you to set the number of decimal places for *dimensions* and *tolerances*.



The **Precision** of *angular dimensions* can be set to *degrees*, *minutes*, and *seconds*.

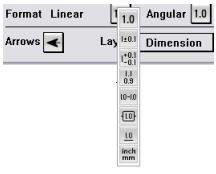
Set the Precision

1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. Set the number of decimal places for *linear* and *angular dimensions* and *tolerances* in the corresponding *pop-up* menus.
3. Click the **Apply** button.
   The specified precisions are valid for all selected and all future dimensions until the end of the current ***DraftBoard*** session.

You can save the selected *precision* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

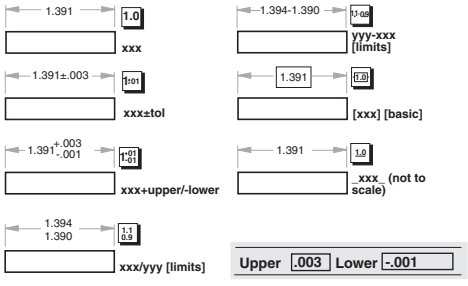
Dimension Formats

Dimension formats including tolerance formats can be specified in the main window of the *Dimension Editor*.

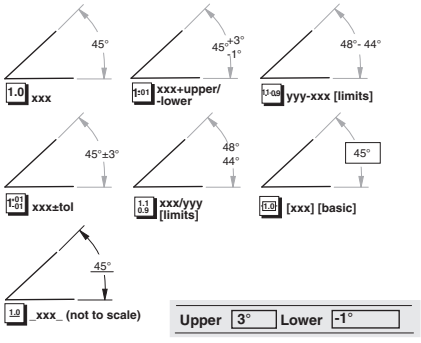


Adding Tolerance Formats

1. Choose **Preferences** in the **Dimension** menu.
   The *Dimension Editor* is displayed.
2. Select *Dimension formats* for *linear* and *angular* dimensions.
   The illustration below shows the appearance of each *Dimension format* for *linear dimensions* if you entered the limits shown in the status line.



   The illustration below shows the appearance of each *tolerance format* for *angular dimensions* if you entered the limits shown in the status line.



3. Click the **Dim Text** button.

   The **Dimension Text** dialog box is displayed.

The default value for the ***DIN*** *standard* is **70%** of the dimension text size.

4. Enter a value for the *tolerance text size* into the **Tol.-Text size** field. This value specifies the *text size for tolerances* in percentage of the *dimension text size*.
5. Click **OK**.

   The **Dimension Text** dialog box is closed.
6. Close the *Dimension Editor* by clicking the **Close** box in the upper left corner of the *Dimension Editor*.
7. Draw a line.
8. Select a linear dimension from the *Dimension Tool palette* such as the **Horizontal Dimension** tool and dimension the line.
9. Enter in the status line values for the *upper* and *lower* tolerance limits.
10. Press the **Enter** key.

    The dimension is redrawn and displays the tolerance limits you specified.

    The specified *tolerance limits* become the default limits until you enter different values while dimensioning.

Entering Text into the Tolerance fields

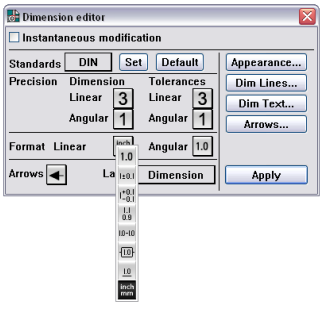
Instead of typing in numbers to calculate the tolerances you can also enter text, such as **H7** in the *upper* and *lower* entry fields of the status line or even leave one of these fields empty. The dimension will reflect exactly what you have entered in the status line.

You must press the **Enter** key to activate the entries made in the Status Line.

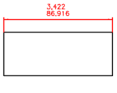
When you enter text in the status line you should use the **upper/ -lower** tolerance format in the *Dimension Editor*.

Dual Dimension Format

For dual dimensions (displaying simultaneously ***mm*** and ***inch*** *units*, independent from the units set in the **Units** dialog window in the **Preferences** submenu in the **Layout** menu) choose the corresponding format in the format list for linear dimensions.



When you select the dimension format **Dual Dimension** in the *Dimension editor*, all dimensions will display simultaneously ***inch*** and ***mm*** values.



Dimension Layer

DraftBoard places dimensions automatically onto that layer you have selected from the **Layer** *pop-up* menu in the *Dimension Editor*.

Placing dimensions automatically on a separate layer

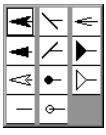
1. Choose **Preferences** in the **Dimension** menu.
2. Select a layer in the **Layer** *pop-up* menu in the *Dimension Editor* where all dimensions should go to.
3. Close the *Dimension Editor* by clicking the **Close** box in the upper right corner of the *Dimension Editor*.
4. Dimension as usual.

   Subsequent dimensions automatically go on the dimension layer rather than the work layer.

For the **DIN** Standard the **Dimension** Layer is set as default layer.

Arrows

The **Arrows** *pop-up* menu allows you to choose one of 11 arrowheads for dimension lines.



Arrowheads for *arrow lines* are defined separately under **Arrows** in the **Pen** menu.

The arrowhead selection for dimensions is valid until the end of the current ***DraftBoard*** session. You can save the selected *arrowhead* permanently with the **Save Preferences** command from the **Preferences** submenu of the **Layout** menu.

Instantaneous Modification

When this option in the *Dimension Editor* is checked all changes made in the *Dimension Editor* take effect immediately on all selected dimensions. The **Apply** button is disabled as long as this option is active.

Apply button

When you click the **Apply** button all modifications made in the *Dimension Editor* are applied to all selected dimensions except the two values in the **Appearance** dialog that are marked with a **#** symbol in front.

All modifications are valid for future dimensions until the end of the current ***DraftBoard*** session. When you want to edit only some selected dimensions you have to select the standard again after you have clicked the **Apply** button to restore its original settings.

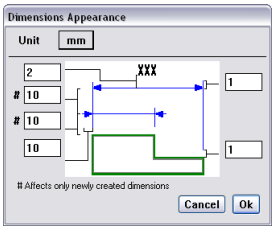
If you want to make these changes permanent you must choose **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

Closing the Dimension Editor

You close the *dimension editor* by clicking the **Close** box in the upper right corner of the *Dimension Editor*. All modifications made in the *Dimension Editor* are valid until the end of the current ***DraftBoard*** session

Appearance

All settings for *Dimension Appearance* are done in the **Appearance** dialog box. When you click the **Appearance** button the following dialog box is displayed:



The **Appearance** dialog box allows the following settings:

Units In this *pop-up* menu you select the units (**mm** or **inch**) for all settings in the **Appearance** dialog box. These units are only valid for the setting in the **Appearance** dialog

box and do not affect the units set for the drawing in the **Preferences** submenu of the **Layout** menu.

Entry fields In these entry boxes you enter the desired offset values for the different elements of a dimension.

When you click the **Apply** button in the *Dimension Editor* the new settings are assigned to the selected dimensions except the two values you entered in the *# fields* which will affect only future dimensions.

#-Symbols Values you enter in the two entry fields which have a ***#*** *Symbol* in front affect only newly created dimensions (not selected dimensions). These two entry fields display values for the perpendicular distance between objects being dimensioned and for the *first* and all further *dimension lines*.

OK When you click the **OK** button the **Appearance** dialog box is closed and all settings are assigned to the current standard.

Modifying the dimension appearance

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid only for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.

If you want to modify the *Appearance* for another *Dimension Standard* you must first change the **Standard** in the *Dimension Editor* and then modify the *Appearance* in the **Appearance** dialog box.

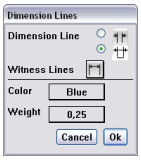
3. ***Drag*** in the **Standard** *pop-up* menu to the standard you want to modify.
4. Click the **Appearance** button in the *Dimension Editor*.
   The **Appearance** dialog box is displayed.
5. Select **mm** or **Inch** in the **Units** *pop-up* menu for all changes you want to make in the **Appearance** dialog box.
6. Modify the *Dimension Appearance* by entering appropriate values in the related entry fields.
7. Click **OK**.
   The **Appearance** dialog box is closed.
8. Click the **Apply** button if you want to assign the new settings to selected dimensions.
   All selected dimensions are redrawn and reflect the new appearance set in the **Appearance** dialog box except the two values you entered in the ***# fields*** which will affect only newly created dimensions.

Click the **Standard** button in the *Dimension Editor* if you want to reset all parameters to its original values.

Dimension Lines

All settings for *Dimension Lines* are done in the **Dimension Lines** dialog box.

When you click the **Dim Lines** button the following dialog box is displayed:



The **Dimension Lines** dialog box allows the following settings:

The **Dimension line** options allow you to choose whether the witness lines are connected by a dimension line (DIN) or not for all dimension text that is not placed between witness lines.

Witness lines In this *pop-up* menu you select the number of witness lines: *left side*, *right side*, *both sides* or *no witness lines*.

Color In this *pop-up* menu you select the *color* for the dimension lines.

Weight In this *pop-up* menu you select the *line weight* for dimension lines. You can choose from the same *line weights* offered in the **Weight** submenu of the **Pen** menu.

If you want to make these changes permanent you must choose **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

OK When you click the **OK** button the **Dimension Lines** dialog box is closed and all settings are assigned to the current standard.

Modifying dimension lines

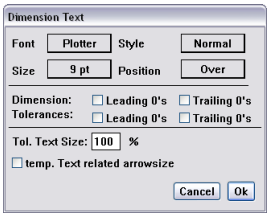
1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid only for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** *pop-up* menu to the standard you want to modify.
4. Click the **DIM Lines** button in the *dimension editor*.
   The **Dimension Lines** dialog box is displayed.
5. Select the all options you want to set for dimension lines.
6. Click **OK**.
   The **Dimension Lines** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

Dimension Text

All settings for *dimension text* (including *Welding* and *Surface Symbols*) are done in the **Dimension Text** dialog box.

All changes made in the **Text** menu have no effect on dimensions.

When you click the **Dim Text** button the following dialog box is displayed:

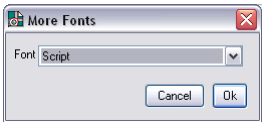


If the option **Instantaneous modifications** in the main window of the *Dimension editor* is checked all modifications done in the **Dimension Text** dialog box take effect immediately.

The **Dimension Text** dialog box allows the following settings:

Font — This *pop-up* menu allows you to choose the *text font* for the selected standard and for all selected and future dimensions.

more — The **Font** submenu can display up to *20 fonts*. If you have installed more than 20 fonts you have to click the **more** button to get access to all available fonts.



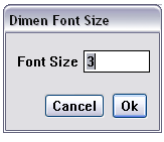
The **Fonts** dialog box lists all fonts installed on your computer including ***DraftBoard's*** plotter fonts.

Whenever you want a font set to exact specifications, use a **Plotter** font, which conforms to ANSI standards, since *PostScript* and *TrueType* fonts are *Proportional Fonts* that can deviate slightly from the specified size. When you scale a drawing using the **Drawing Size** command and having the options *keep text size* and *keep dimension text size* not checked, you have to select all *dimensions* and assign their text size in the *Dimension Editor* again.

Size — This *pop-up* menu allows you to choose the *text size* for the selected standard and for all selected and future dimensions.

The *text size* can be specified either in *point*, in *mm* or *inch* depending on the units you have set under **Units** in the **Preferences** submenu of the **Layout** menu.

Other — Choosing **Other** from the **Size** submenu brings up a dialog box allowing you to specify a nonstandard *text size* in the units you've set for your drawing. You can specify point size by entering **pt** after the value in the entry box; for example, if you enter **11 pt** the font size will be *11 point*.



Style — This *pop-up* menu allows you to choose the *text style* for the selected standard and for all selected and future dimensions.

Leading 0 Determines whether or not the zero to the left of the decimal point appears.

Trailing 0's Determines whether or not zeros to the right of the decimal point are displayed. If set, the precision determines the number of trailing zeros.

Tol. Text size In this entry field you specify the *text size for tolerances* as a percentage of the *dimension text size*.

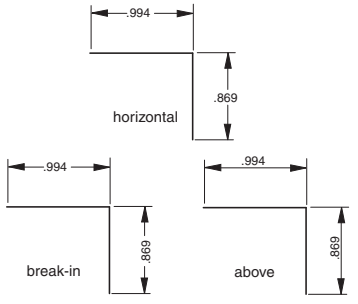
temp. Text related arrow size When this option is checked the ***arrow size*** of the *dimension lines* will be adjusted when the *text size* is modified. The default arrow size for dimension lines is normally only increased or reduced while scaling a drawing. Using very large or small dimension text makes it sometimes necessary to adjust the *arrow size* to the selected *text size*.

This option is deactivated by default and can be used only for selected dimensions using the **Apply** button.

Position This *pop-up* menu allows you to set the *position* of *dimension text* for the selected standard and for all selected and future dimensions. You can choose from the following options:

- *horizontal*
- *over*
- *break-in*
- *under*

The first option determines if the dimension text is always displayed *horizontally*. The following three options determine if the dimension text is placed above (*over*) in between (*break-in*) or below (*under*) the dimension line.



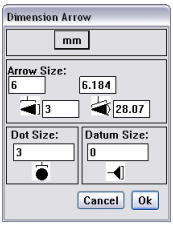
Modifying dimension text

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** pop-up menu to the standard you want to modify.
4. Click the **DIM Text** button in the *Dimension Editor*.
   The **Dimension Text** dialog box is displayed.
5. Select all the options you want to set for dimension text.
6. Click **OK**.
   The **Dimension Text** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

Arrow Size

All settings for *Dimension Arrows* are done in the **Arrow Size** dialog box.

When you click the **Arrow Size** button the following dialog box is displayed:



The **Arrows** dialog box allows the following settings:

Units In this *pop-up* menu you select the units (**mm** or **inch**) for all settings in the **Arrows** dialog box. These units are only valid for the setting in the **Arrows** dialog box and do not affect the units you set for your drawing in the **Preferences** submenu in the **Layout** menu.

Arrows Size In these entry boxes you can set the size of the arrowhead either by specifying its *length/height ratio* (ANSI) or setting values for the *length of the edge* and the *angle of the tip* of the arrowhead (DIN).

Dot Size This value describes the diameter of *circular* and *slash* arrowheads.

Setting the Arrow Size

1. Select all dimensions you want to modify. If no dimensions are selected all changes are valid for future dimensions.
2. Choose **Preferences** in the **Dimension** menu to display the *Dimension Editor*.
3. ***Drag*** in the **Standard** *pop-up menu* to the standard you want to modify.
4. Click the **Arrow Size** button in the *Dimension Editor*.
   The **Arrow Size** dialog box is displayed.
5. Select all the options you want to set for the size of the arrowheads.
6. Click **OK**.
   The **Arrow Size** dialog box is closed.
7. Click the **Apply** button if you want to assign the new settings to a selected dimension.
   All selected dimensions are redrawn and reflect the new appearance set in the *dimension editor*.

CHAPTER

10

Basic Drafting

Zooming

Screen Redraws

The Drafting Process

Drawing At Full Scale

Paper Size and Drawing Scale

Scaling Drawings

Scaling Commands

Printing or Plotting a Drawing

Basic Drafting

This chapter describes techniques for *Printing* and *Scaling*. There are two different ways to change the drawing size:

- *Zooming*
  You can change the magnification of your drawing by zooming in and out using commands, zoom tools, and strokes. Zooming changes the visual size of the drawing, never the physical size of the objects.
- *Scaling*
  Scaling means to set the visual size of the drawing to fit to the selected paper size when printing or plotting. Scaling too changes the visual size of the drawing, never the physical size of the objects.

This chapter describes the following techniques:

- Zooming
- Full-scale drawing
- How scaling affects *text*, *dimensions*, *line patterns* and *hatch patterns*
- Scaling commands
- Printing or plotting drawings on standard paper sizes

Most drafters begin a project by deciding how to scale the part first. Using ***DraftBoard*** you create all parts at full scale (**1:1**).

You never physically scale the part, instead you set the visual scale of the geometry (the visual size of the drawing) to fit to the selected paper size when printing or plotting the drawing.

Zooming

DraftBoard gives you several ways to *zoom in* or *zoom out* on your drawing.

Zoom Commands

You find four *Zoom commands* in the **Arrange** menu: **Zoom In**, **Zoom Out**, **Zoom All** and **Zoom Previous**. Choosing the **Zoom Previous** command takes you to the last magnification. If you want to zoom a particular area, use the **Stroke** feature or the *Magnifying* tool for zooming. *Strokes* are described later in this chapter.

***Important**: Each Zooming is followed by a screen redraw. You can press the **Pause** or **Esc** key to stop redrawing the screen.*

Zoom All Ctrl+F

This command in the **Arrange** menu zooms in or out to make all parts of the drawing visible on the screen.

Zoom In Ctrl+]

This command in the **Arrange** menu zooms in on the center of the screen by a factor of two. You cannot specify a particular area for enlargement.

Zoom Out Ctrl+[

This command in the **Arrange** menu zooms out from the center of the screen by a factor of two. You cannot specify the area of reduction.

Zoom Previous

This command in the **Arrange** menu zooms to the previous magnification, as many as five times.

Zoom Tools

Select the appropriate zoom tool from the **View Control** tool palette.

***Important**: Each Zooming is followed by a screen redraw. You can press the **Pause** or **Esc** key to stop redrawing the screen.*

Using the magnifying glass tool

With the **Zoom** tools you ***drag*** a box around an area on the screen, so only that area is magnified on the screen.

1. Select a **Zoom Tool** from the tool palette.
2. With the left mouse button pressed ***drag*** a box around an area on the screen you want to magnify or reduce.
3. Release the mouse button.

   The content of the ***dragged*** box is made visible on the screen.

Zoom In Tool

This tool zooms in by the specified factor. The default factor is **2**. This is a visual rather than a physical change.

To zoom into or out of a drawing you can use as well the mouse wheel, if your mouse is supporting this function.

When you click in the drawing area, that position is redisplayed in the center of the screen, and the drawing is enlarged by a factor of **2**.

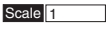
You can also ***drag*** a box around an area, so only that area is magnified.

The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press **Enter**. Pressing the **Ctrl** key while using this tool causes a **Zoom Out**.

Zoom Out Tool

This tool zooms out by the specified factor. The default factor is **0.5.** This is a visual rather than a physical change.

When you click in the drawing area, that position is displayed in the center of the screen and the drawing is reduced by a factor of **0.5**.



The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press **Enter**.

Pressing the **Ctrl** key while using this tool causes a **Zoom In**.

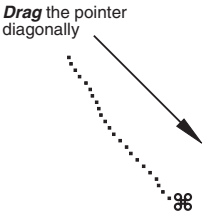
Zoom Stroke

You can use stroke commands to zoom—magnify or reduce your drawing. *Stroke commands* are useful because they don't require you to get out of the tool you're using to zoom. Hold down the **Ctrl** and **Shift** key and ***drag*** the pointer that appears diagonally across the screen as described below. The pointer takes on the shape shown below when you hold down the **Ctrl** and **Shift** key .

Using Stroke zoom

Depending of the stroke direction you can execute different *Stroke commands*:

Note: *For **Zoom In** and **Zoom Out**, the size and location of the stroke rectangle is important for determining the result of the **Zoom** operation. For **Zoom Previous**, the size and location of the stroke rectangle is irrelevant. All cases just give the previous magnification.*



Drag **Diagonally**	**Result**
Upper left to lower right	Zoom-in enlargement centered over the stroked area.
Lower right to upper left	Reverses **Zoom In** stroke to the previous magnification.
Upper right to lower left	Zoom-out reduction-the current screen reduces to the size of the area defined by the stroke.
Lower left to upper right	Reverses **Zoom Out** stroke to previous magnification.

Panning Screen Contents

The tool **Pan Drawing area** allows to move the content of a screen. You find this tool in the **Views** subpalette.

Pan Drawing area

This tool displays a hand icon that allows you to move the screen area while the mouse button is pressed.

The status line contains no boxes.

Panning a drawing area

1. Select the **Pan drawing area** tool in the **Views** subpalette of the *Tool Palette*.
2. Press the mouse button and ***move*** the content of the screen while the mouse button is pressed.

Screen Appearance

DraftBoard offers several commands to control screen appearance.

Redraw Screen Ctrl+R

This command in the **Arrange** menu refreshes the screen. When you make changes to your constructions, the geometry may not be redrawn cleanly in the drawing area.

Redrawing the screen

To redraw all of the geometry and remove extraneous geometry and points, choose the **Redraw Screen** command from the **Arrange** menu.

Stopping a screen refresh

You can press the **Esc** or **Break** key to stop the redrawing of the screen.

Important*: You should use the* ***Break*** *key instead of the* ***Esc*** *key, since Windows displays the Task manager when the* ***Redraw screen*** *command was initiated by the key combination* ***Ctrl+R****.*

Invert

This command in the **Preferences** submenu in the **Layout** menu allows you to switch the color of the drawing area from *White* to *Black* and vice versa. The default color is *White*.

The Drafting Process

When creating a drawing you have to consider several items to guarantee that the drawing is printed within a *standard drawing format* according to the *standard drawing rules*. While you don't have to follow the these suggestions in order, you may want to use the following scheme until you have enough experience to know how each step affects the overall project.

Creating a drawing

1. Start ***DraftBoard***.

   DraftBoard automatically displays an **Untitled** document.
2. If necessary, set the **Units** of measure you want to use with the **Preferences** command in the **Layout** menu.
3. Use the **Drawing Size** command in the **Layout** menu to set the drafting scale to **1:1**.
4. Construct the geometry at full size, saving the construction as you work. Use the **Zoom** commands and tools to display the part at an appropriate magnification so that you can see what you are doing.
5. Use the **Print Setup** command in the **File** menu to set the paper size you will use for plotting or printing.

The precision you specify for the selected unit defines only the number of digits you can enter in the status line, not the precision of the object geometry, that is always saved with double precision. Therefore you can change the precision at any time during the drafting process.

6. Use the **Drawing Size** command in the **Layout** menu to set the visual scale of your drawing to fit on the specified paper size for plotting or printing.

   You may specify the drafting scale, such as **1:100** or **1cm:1m** for **1** centimeter to represent **1** meter.

   If you are not concerned about scale and you only want the final geometry to fit on the selected paper size, you can specify **Fit** to adjust the geometry automatically to the paper size.

7. *Dimension* the part and *add text*.

8. Import a *standard drawing format* with the **Import** command in the **File** menu from the **Layout** folder.

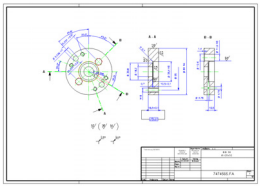
9. With the **Preview Layout** command in the **File** menu you can check size and orientation of the printing area that is displayed by a grey border to make final adjustment if necessary.

10. Choose **Print** from the **File** menu.

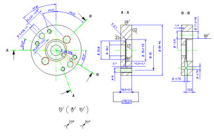
    The drawing is printed or plotted on the specified printer or plotter. *Text* and *dimensions* are printed in the size you specified, since they were added **after** you scaled the drawing.

    When you have set for example the size for *text* and *dimensions* to **3,5 mm**, *text* and *dimensions* will be printed in a size of **3,5 mm**.

How you have to proceed if you added *text* and *dimensions* **before** you scaled the drawing is described in the following sections of this chapter.



Flange plotted with a standard drawing format including block and title.



Flange plotted without a standard drawing format.

The above described procedure recommends to scale the drawing (to adjust the visual scale to the specified paper size) **before** you add *text* and *dimensions*.

Of course you can scale drawings in ***DraftBoard*** after you have added *text* and *dimensions*. But that requires that you understand the different techniques ***DraftBoard*** uses for scaling. These procedures are described in the following sections of this chapter.

10

Drawing at Full Scale

In ***DraftBoard*** you create all parts at its actual size. It doesn't matter whether you design a large industry building or a small detailed blueprint.

When printing you adjust the visual scale (the view of the drawing) to fit the paper you will use. The true size of all objects saved in the ***DraftBoard*** *Database* is never affected by scaling.

DraftBoard saves all objects at their true measurements, regardless of which drawing scale you specify with the **Drawing Size** command and to what extend you magnified the drawing with the *Zoom commands*.

With these commands you only change the visual size of the objects on the screen or on the paper but never its true size saved in the ***DraftBoard*** *database*.

In this way the part dimensions to its true-to-life measurements regardless how they are displayed on the screen or printed on the paper.

Drawing at full scale has the following advantages:

- scaling mistakes are eliminated
- dimensions are automatic (you must dimension manually if you do not draw at full scale)

- associative dimensions update when the object is edited (manual dimensions do not)
- the size relationship of imported parts is compatible.
- calculations for *2D analysis* are accurate

Creating and editing objects at full size

When you open a new ***DraftBoard*** document, the drawing area is a sheet that is infinitely large so that you can design anything at full size. For a simplistic example, here's how to draw and view a line *24 meters* long:

1. Draw a line.
2. For the Length of the line type **24 m** in the **Length** field of the status line.

   The line extends off the screen.
3. Choose **Zoom All** from the **Arrange** menu.

   The entire **24** meter line is visible on the screen.

Zoom All magnifies or reduces all objects on your drawing to fill the screen—regardless of how big or small the objects are.

While zooming ***DraftBoard*** calculates the scale required to display the line at its full length on the screen. When you zoom again ***DraftBoard*** doesn't use the scaled length to calculate the new scale (this would lead to scaling mistakes the more often you scale) but its original true length saved in the ***DraftBoard*** *database*. The *Zooming commands* allow you to create accurate full-scale drawings and display them at the magnification required.

Paper Size and Drawing Scale

Before you can check if your drawing can be printed at full size on a specific paper format you have to specify a paper size and orientation for your installed printer or plotter.

Specifying the paper size

To specify a paper size you choose the **Print setup** command.

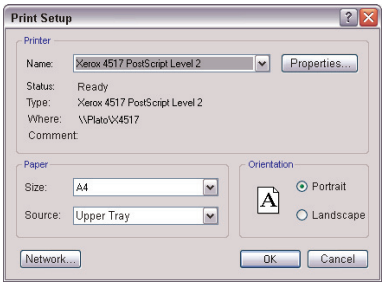
Print Setup

This command in the **File** menu allows you to select a printer, set the page size, orientation, and other options.

Setting up the page

1. Select **Print Setup** in the **File** menu.

   The following the dialog box is displayed.



2. Choose your printer in the **Specific Printer** field if it is not installed as the **Default Printer**.
3. Select the paper size from the **Paper Size** *pop-up* menu.
4. Select the paper source from the **Paper Source** *pop-up* menu
5. Specify the **Orientation** (*Portrait* or *Landscape*).
6. Click the **Option** button.

   Specify in the displayed dialog box if you want to print your drawing to a printer or save it to a file.
7. Click the **OK** button.

In case your printer is not listed in the list of available printers you must first install your printer under Windows. A detailed description how to install a printer under *Windows* can be found in the *Windows Reference Manual*.

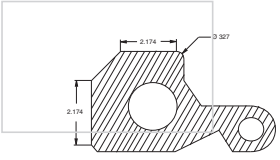
Checking the View Scale

After you have specified the *paper format* you must check if the objects of your drawing are too large or too small in relation to the selected paper format and if its view has to be scaled to fit on the specified paper size.

The **Preview Layout** command in the **File** menu allows you to check the size ratio between the objects and the specified paper size.

Preview Layout

This command in the **File** menu displays the *printing area* of the selected paper at its true size and selected orientation. If you have not specified a *printing/plotting size*, the default size is used.



If the drawing does not fit into the displayed rectangle representing the printing area of the selected paper size, you must scale the drawing.

Printing/Plotting region

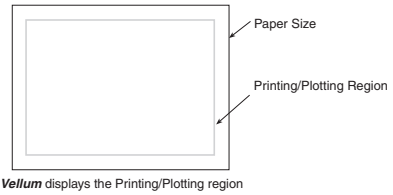
If the grey boundary representing the page size is not visible after scaling choose **Zoom all** while the **Drawing Size** dialog box is displayed.

The *printing/plotting region* is displayed when choosing the **Preview Layout** command and the **Drawing size** command.

With both commands a grey rectangle appears in the drawing area. This rectangle represents the *printing/plotting region* of the page so you can see how your drawing fits the paper size.

The *printing/plotting region* is displayed filling the screen with the **Origin** (**0,0**) placed in the center of the page. If you cannot see your drawing on the screen you select the **Zoom All** command in the **Arrange** menu.

The *plotting region* is smaller than the actual *page size* because most printers and plotters cannot plot to the edge of the paper. If you want to see the plotable outline all the time, click the **Always Display** button.



Vellum displays the Printing/Plotting region

When the boundary of the printing/plotting region appears, you can ***drag*** the boundary rectangle around the drawing area.

When you choose the **Preview Layout** command again or select any tool from the tool palette the *Printing/plotting region* disappears.

By the size relationship between the existing object geometry and the displayed *printing/plotting region* you can check visually if you have to scale the view of your drawing. To do so you have to know how the different elements of a drawing such as *geometry*, *text*, *line patterns* behave while scaling, this described in the next section.

10

Scaling Drawings

Whenever the created object geometry is too large or too small in relation to the specified paper format drawings must be scaled.

Scaling affects elements of a drawing differently. While scaling ***DraftBoard*** distinguishes between the following elements:

- object geometry
- hatch patterns
- line patterns
- text
- dimensions

For scaling you use the **Drawing Size** command in the **Layout** menu.

Scaling Object geometry

Let's assume that a drawing exists only of object geometry created with solid lines.

If the grey boundary representing the page size is not visible after scaling choose **Zoom all** while the **Drawing Size** dialog box is displayed..

In that case you can see the result of the scaling only by the changed size of the paper format represented by a grey rectangle in the drawing area for the *printing/plotting region* of the page.

If the grey rectangle for the page size isn't visible you can't see the reduced or increased size of the object geometry caused by the scaling

command, since the size of the individual objects didn't change in relation to each other but only in relation to the paper size.

Scaling Line patterns and Crosshatch patterns

When you use *hatches* and *dashed lines* in a drawing, the line dashes and hatch patterns are displayed at their true size.

When these *line dashes* are very small in relation to the remaining object geometry (size of *object geometry* in meters and size of *line dashes* in millimeters) you can't identify these *line dashes* on the screen before scaling, since they are displayed at actual size.

When you scale the drawing the object geometry is scaled while the dashes of the lines and hatch patterns keep their true size. Therefore *line and hatch patterns* change size in relation to each other.

When you reduce the view scale of the drawing the size of the object geometry is reduced in relation to the dashed lines with the result that *line* and *hatch patterns* are now visible on the screen and on a print out.

In case the *dashed* or *dotted line patterns* are not displayed correctly because the scale factor is too large or too small you can adjust the line pattern using **Edit Pattern** the command in the **Pen** menu.

Remember: Lines with *dashed* or *dotted line patterns* and *Hatch patterns* are never scalded in ***DraftBoard*** but always displayed at their true size.

Scaling Text and Dimensions

If your unscaled construction is very large or very small compared to the size of added *text* and *dimensions* you have to scale the view of the drawing.

While scaling ***DraftBoard*** distinguishes if the options

- **Keep Text size**
- **Keep Dimension Text size**

are checked or not. When both options are checked *text* and *dimensions* are **not** scaled and keep the size you have assigned in the **Text** menu or in the **Dimension Editor**. With these options checked the size of the object geometry will be increased or reduced in relation to *text* and *dimensions* .

However when you scale a drawing without having these options checked

- *text* and *dimensions* are scaled
- *text* and *dimensions* that you add **after** scaling are not scaled and displayed at true size.

This feature allows you to scale *text* and *dimensions* if necessary.

Options Keep Text size and Keep Dimension Text size

The options **Keep Text size** and **Keep Dimension Text** size is only available with the **Drawing Size** command.

Remember: ***DraftBoard*** scales *existing* text and dimensions with the same scale factor as the object geometry and keeps *text size* and *dimension text size* only for future text and dimensions which are added after scaling.

Text and *dimensions* you place **after** scaling will be displayed at true size even if the options *Keep Text Size* and *Dimension Text Size* are active.

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

Plotter Fonts

When you are using a plotter, and the text should be displayed at its accurate size, you should specify a plotter font for text and dimensions on your drawing, since all other fonts are proportional and may differ in their appearance slightly from the assigned size.

You can also specify different text styles (such as *italic* or *bold)* for plotter fonts in the **Text** menu and generate special characters and accents as described in the *Appendix* of this manual.

All *Plotter Fonts* available with ***DraftBoard*** are listed in *Appendix* ***A*** of this *Reference Guide.*

Scaling Rules

While scaling the different elements of a drawing behave as follows:

- *object geometry* is **always** scaled
- *line* and *hatch patterns* are **never** scaled
- *text* and *dimensions* are **not** scaled if the options *keep text size* and *keep dimension text size* are active.
- *text* and *dimensions* are **always** scaled if the options *keep text size* and *keep dimension text size* are **not** checked.

To keep *text* and *dimensions* at a constant size while scaling and having the options *keep text size* and *keep dimension text size* not checked you have to:

- add *text* and *dimensions* **after** scaling

  or

- you must select all *text* and *dimensions* and assign their text size again.

It is advisable to use the **Selection mask** to select all text and dimensions in case you have to assign the accurate text size **after** scaling.

Scaling Commands

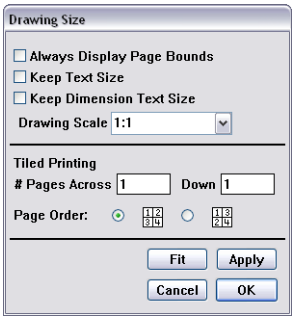
If your unscaled construction is very large or very small compared to the size of the specified page format you have to adjust the view scale of your drawing to fit the selected paper size. To do this you use the **Drawing Size** command in the **Layout** menu.

Drawing Size

This command in the **Layout** menu allows you to see the size of the maximum *plotting/printing* area relative to the drawing and to specify the scaling of the drawing so it fits the paper size and orientation set by the **Print Setup** command.

Before you choose the **Drawing Size** command in the **Layout** menu, you should use *zoom out* until your drawing fills half the screen size.

When you choose the **Drawing Size** command the following dialog box is displayed:



The **Drawing Size** dialog box allows the following settings:

Always display Page Bounds — When you check this option the rectangle for the *printing/plotting region* is always displayed at its true size in the drawing area even if the **Drawing Size** dialog box is closed.

Keep Text Size — When this option is checked *text* is **not** scaled but keeps the *text size* set before scaling.

Keep Dimension Text Size — When this option is checked *dimensions* are **not** scaled but keep the *text size* set before scaling under **Preferences** in the **Dimension** menu.

Drawing Scale — In the **Scale** box you enter a *scale factor* for your drawing specifying the ratio between the *printed* and the *true size* of your drawing.

To specify a scale you can press the arrow beside the **Scale** box and ***drag*** to one of the standard ratios from the menu or enter real values, such as **1cm: 1m**, which is converted to **1:100** by ***DraftBoard***.

Fit — When you click this button the drawing is scaled to fit the printing region of the specified paper size. The calculated scale factor is displayed in the **Scale** box and the **Drawing Size** dialog box stays open.

Apply — When you click the **Apply** button the drawing is scaled according to the selected factor and you can see the specified scaling on the drawing area. The **Drawing Size** dialog box stays open.

OK — When you click the **OK** button the drawing is scaled according to the specified scale factor. The **Drawing Size** dialog box is closed and

the rectangle of the printing region disappears on the screen when the option **always display Page Bounds** is not checked.

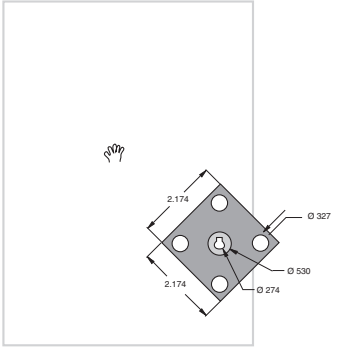
Setting up the drawing size using the Drawing Size command

1. Choose **Drawing Size** from the **Layout** menu.

   The **Drawing Size** dialog box is displayed.

   With the dialog the grey rectangle for the *printing/plotting region* is displayed at its true size on the drawing area. When the boundary of the *printing/plotting region* appears, you can ***drag*** the boundary rectangle around the drawing area.

The **Cancel** Button on the **Drawing Size** dialog box doesn't abort changes if a **Fit** or **Apply** are applied first.



If you want to change the *Page orientation* of the displayed page boundary you have to use the **Print Setup** Command.

2. Specify the scale ratio.

   You can press the arrow beside the **Scale** box and ***drag*** to one of the standard ratios from the menu or double-click the **Scale** box and enter a scale (the ratio) you want to use. You can enter real values, such as **1cm:1m**, which is converted to **1:100** by ***DraftBoard***.

   If you don't need a specific scale, you can click the **Fit** button to scale the drawing to fit the paper.

3. When *text* and *dimensions* should keep their true size choose the options **Keep Text Size** and **Keep Dimension Text Size**. If you don't select these options you have to adjust *text* and *dimensions* **after** scaling.

4. When you want to control the scaling on the screen click the **Apply** button.

   The drawing is scaled and the **Drawing Size** dialog box remains open. The geometry itself does not change scale; it only changes visually, not physically.

5. If you want to print multiple sheets and tape them together, follow the directions for *Tiled Printing* described later in this chapter.

6. Click **OK**.

Printing or Plotting a Drawing

DraftBoard prints and plots on most printers and plotters supported by your computer. After you have followed the manufacturer's instructions for installing and setting up the printer or plotter, you can size your drawing for the paper you will be using.

Print Ctrl+P

This command in the **File** menu prints or plots the current document as specified in **Print Setup** .

The **Print Setup** command allows to change the current plotter or printer and to set options such as *Paper Size* and *Paper Orientation*.

The area printed or plotted is the portion that fits on the page size when the origin (**0,0**) is placed in the center of the page. Use the **Drawing Size** command from the **Layout** menu to reposition the *print/plot region*.

By repositioning the *printing/plotting region* it is possible to print only part of a drawing, since the only part that is printed is the part that lies within the printing region.

Plotting to a file

If you want to use ***DraftBoard*** *drawings in a Text Editor or Layout program the* ***EPS*** *file format is recommended, since it offers the best quality and allows to scale the graphic after placing without any quality loss. How to create an* ***EPS*** *file is described in Chapter 11 DraftBoard Documents.*

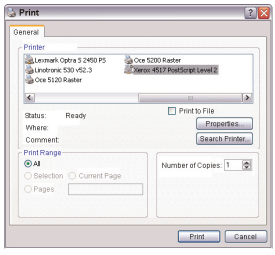
You can plot to a file rather than to a plotter or printer. In that way, you don't have to have a plotter attached to your computer, you can have someone else plot the drawing without having a copy of ***DraftBoard*** on the plotter's computer.

The type of plotter you choose when you are setting up the page determines the format of the plot file. For example, if you choose a **PostScript** printer, the file format will be **E**ncapsulated **P**ost**S**cript; and the **HPGL** language is used when you select **Hewlett Packard** plotters. The computer that finally plots the file must have an application compatible with the file format of your printer or plotter.

Plotting to a file

1. Choose **Print** command in the **File** menu.

   The **Print** dialog box is displayed.



2. Mark the **Print to file** option.
3. Click **OK**.

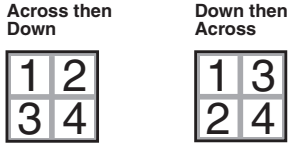
   The **Print to file** dialog box is displayed.
4. Enter a name for the print file.
5. Click **OK**.

   The drawing will be printed to file and saved under the name you specified.

Tiled Printing

If you are using a printer rather than a plotter, you may want to tape pages together to get a larger drawing than laser printers allow. Specify the layout for those pages by choosing **Drawing Size** from the **Layout** menu.

1. Choose **Drawing Size** from the **Layout** menu.
2. Specify the number of pages *across* for the final drawing.
3. Specify the number of pages *down* for the final drawing.
4. Specify the order for printing the pages (*across then down* or *down then across*).



5. Click the **Fit** button.

   The drawing will be scaled to fit to the number of pages you specified.
6. With the mouse button pressed reposition the displayed rectangles representing the number of pages you specified to cover these objects you want to print.
7. Click **OK**.

   Your drawing is scaled to fit the number of pages and layout, as you have specified.

CHAPTER

11

Structuring Drawings

Layer

Associative Views

Drawing Formats

Structuring Drawings

DraftBoard provides the following features to structure drawings:

- Layers
- Associative Detail Views

These tools allow you to use ***DraftBoard*** individually for your design tasks.

Layers

Layers are particularly useful in helping you view and plot complex drawings. For example, when you dimension a part, the dimensions can be placed on a separate layer which you can display or not, as your needs require. *Layers* allow you to plot different versions of the same document for quick and easy specialized blueprints.

Layers are often described as transparent films. But actual they are just another method to group objects.

- You can hide the *dimension layer* to exhibit the idea of a design to a planning team and show the dimensions when presenting the drawing to engineers.
- You can hide some drawing components when you plot. For example, you can hide the construction layer so that construction lines and geometry don't plot, but they remain in the drawing ready for use when you want to alter it.
- You can construct different layouts using one layer as the basis. For example, you can use a floor plan and then construct the electrical and plumbing plans on different layers. In that way, you can turn off one layer (the electrical plan, for instance) and print the other (the plumbing plan) with the floor plan.

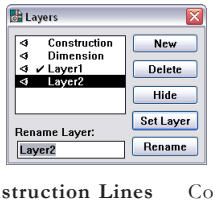
DraftBoard allows you to have up to **256** visible or hidden layers in your drawing. Layer names are limited to **79** characters.

Since the ***DXF*** *format* supports only **31** characters you should limit layer names to **31** characters to guarantee a smooth ***DXF*** *Export.*

Layers Ctrl+L

This command in the **Layout** menu allows you to *create*, *delete*, and *rename* layers, as well as set layer specifications. The *work layer* is the active layer—the layer on which the current construction is created.

The visible layers are indicated in the list box by a **checkmark** and a **dot** indicates the work layer.



DraftBoard includes a **Construction layer** and a **Dimension layer** by default with each drawing.

Construction Lines Construction lines automatically go on the construction layer.

Dimensions If you create a dimension layer, all dimensions added after you create the layer go on the dimension layer.

Note: *The* ***Construction*** *layer will automatically accept all* ***Construction lines*** *created with stroke commands or with the* ***Construction*** *dialog box. All lines of "construction" line type created with the* ***Single Line*** *tool will go onto the current work layer.*

You can edit any geometry or text that is visible, regardless of which layer it is on. If you want to make some geometry unselectable but still visible, use the **Selection Mask** command in the **Edit** menu.

You can use layers for complex drawings so you can hide some objects to simplify the drawing visually or print only specific parts of the drawing. You can think of visible layers as transparent pages and hidden layers as invisible pages. Although you cannot see objects on hidden layers, they do exist. The layer must be visible for you to select objects on the layer. For example, if there are objects on a hidden layer and you choose **Select All**, then delete all objects, those objects on the hidden layer are not deleted.

You can also specify the selectability of layers with the **Selection Mask** command in the **Edit** menu.

Creating a new layer

You can also type a name and click **New**.

Click **New** to create another layer. You can name the layer by typing the name in the **Rename** entry box and clicking **Rename**. You can have as many as **256** layers.

Renaming a layer

Layer names are limited to **79** characters. Since the ***DXF*** *format* supports only **31** characters you should limit layer names to **31** characters to guarantee a smooth ***DXF*** *Export.*

1. Select the name of the layer from the list box.
2. Type the new name.
3. Click the **Rename** button.

Deleting a layer

Note: *You cannot hide or delete a layer that is the current (active) work layer.*

1. Select the name of the layer from the list box.
2. Click the **Delete** button.

   Caution*: The layer and everything on that layer is deleted; however, a warning appears before this happens.*

Making a layer the work layer

1. Select the name from the list.
2. Click the **Set Work** button.

   The layer must be visible before you can make it the work layer.

Using the layer box to specify a different work layer

You can use the **pop-up** layer box at the lower left of the drawing area to specify a different layer as the work layer.

1. Press the mouse button on the box and the menu displays.
2. ***Drag*** to the layer you want to be the work layer and release the mouse button.

Hiding a layer

You can't hide the work layer.

1. Click the layer(s) you want to hide in the list.
2. Click the **Hide** button.

   The visible mark beside the layer name disappears.

Displaying a hidden layer

1. Click the layer(s) you want to make visible in the list.
2. Click the **Show** button.

   A visible mark appears next to the layer's name in the list.

Determining how many objects are on a layer

1. Choose **Selection Mask** from the **Edit** menu.
2. Select the desired layer from those listed in the center box.
3. Choose **Select All** from the **Edit** menu.
4. Choose **Edit Objects** from the **Edit** Menu.

   The number of objects is listed at the top of the dialog box.
5. Close the **Selection Mask** and **Edit Objects** dialog boxes.

Using colors with layers

Layers facilitate color specific wiring diagrams or electrical and plumbing blueprints.

Many CAD drawings show different components in different colors. When you specify a different color for a layer, the layer ***remembers*** the color so that if you add geometry to a layer later, the new geometry appears in the same color as the last geometry constructed on that layer. In this way, you can use the **Visible** pen in **red** on one layer, **green** on another layer, and so on.

1. Select **Layers** from the **Layout** menu.

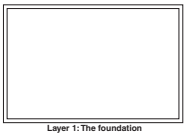
Note: *Layers can be saved as Preferences along with specified colors.*

2. Create a new layer and rename it **Outline green**, indicating you always want to use the pen style **Outline** with a green color on this layer.

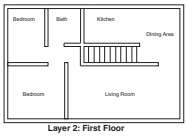
3. Make the layer **Outline green** current.
4. Select **Outline** in the **Style** submenu from the **Pen** menu.
5. Change the pen color from **Black** to **Green** by selecting **Green** in the **Color** submenu from the **Pen** menu.
6. Draw a circle.
   The circle is created with the green color.
7. Select the **Single line** tool.
8. Make **Layer 1** current.
9. Draw a line
   The line is created in black (the default color for the **Outline** pen style).
10. Select the **Rectangle** tool.
11. Make Layer **Outline green** current and draw a rectangle.
    The rectangle is created in **green**.

A Layer Example

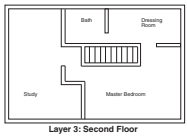
Layers are like transparent sheets that you can turn on and off. This feature is particularly useful for designs with several variations of a component. A common use of layers is to draw the different floor plans and plumbing and wiring plans for a building on different layers. The following examples show simplified drawings using layers.



Layer 1: The foundation



Layer 2: First Floor



Layer 3: Second Floor

In order to print or plot the layers, simply use the **Layers** dialog box to turn on the layers you want to print and turn off all other layers. When you can see what you want to print on the screen, choose **Print** from the **File** menu.

Using Smart Walls with layers

You can also observe how smart walls function in these examples. The interior walls on **Layers 2** and **3** overlay the exterior walls of the foundation.

*Smart walls merge only on same layers; therefore in our example interior walls are not merged with exterior walls and walls on **Layer 2** not with walls on **Layer 3**.*

Associative Views

Views allow you to enlarge or reduce all or part of the geometry for both viewing and printing. This visual change of the view does not affect the actual measurement of the geometry (the model). There exist two types of views in ***DraftBoard***: the **Sheet View** and **Detail Views**.

Sheet View The *Sheet View* shows the object geometry on the sheet at the scale you set with the **Drawing Size** command in the **Layout** menu. The *Sheet View* is the default view when you launch ***DraftBoard***. The *Sheet View* is picked up by the *Sheet camera* and projected on the current sheet. The *Sheet View* is infinite. It has no boundaries.

Detail Views A detail view is created with either the **Detail View** tool on the tool palette. **Detail Views** are always displayed in a view window.

These two different views are easily to identify on the screen:

- Each item displayed **inside** a view window is a **Detail View**.
- The area **outside** (and underneath) all detail view windows is the **Sheet View**.

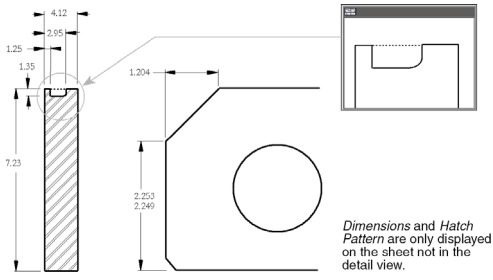
Creating Detail views using the Detail View tool

DraftBoard allows you to create *associative Detail Views*. These *Detail Views* are always placed in a view window and are associative so that when you modify geometry in any view, all views of that geometry reflect the change. There are three deliberate exceptions to this associativity, in line with standard drafting practice:

- Dimensions
- Crosshatching
- Text

They are *view dependant* and appear only in the view in which they were created.

In the illustration below, the detail view shows a **2 : 1** enlargement of the side notch. *Dimensions*, *text* and *hatches* are displayed only on the sheet where they were created.



Regardless of the method you use to create a **View Window**, they all behave the same way.

View Windows will be created in ***DraftBoard*** manually with the **Detail View** tool.

Detail View Tool

How to create a *non-associative* detail view is described at the end of this chapter.

The **Detail View** tool on the **View Control** subpalette of the *Tool Palette* creates a detail view of the designated area of your drawing. The detail view is *associative* so that when you modify the geometry, the detail view reflects the change

Creating an associative detail view

1. Select the **Detail View** icon from the **View Control** subpalette.
2. Enter a **Scale** for your detail view in the status line.

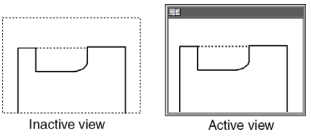
You may also create the rectangle by clicking to place the diagonally-opposite corners.

3. Use the pointer to ***drag*** a rectangle around the area of your drawing that you want placed in the detail view. This rectangle becomes the window frame.
4. Position the pointer in the center of the detail view window frame and ***drag*** the window to a clear area on your drawing.

Activating/deactivating detail views

A *Detail View* is made *active* by clicking inside the view window. An active detail view displays its title bar and **Detail View** pull-down menu.

When a view window displays a title bar the view window is active. When the view window has no title bar, the underlying sheet is active.



A view must be active for you to work in it. Either a detail view is active or the sheet view is active, but only one can be active at a time.

You deactivate a *Detail View* by clicking in the drawing area away from all detail views. In order to draw on the sheet it must be active.

Flatten all Views

***Note**: All geometry invisible will not be projected but cut and deleted. Therefore you must use the **Zoom all** command if you want to project all geometry in a view window.*

The **Flatten all Views** command in the **Window** menu places all geometry visible in a view window with the current scale factor onto the current sheet and removes overlapping lines of equal length and all lines parallel to the view direction. The object geometry is no longer associative to the related model. When you edit a flattened view the related model remains unchanged and vice versa.

In addition all hidden three-dimensional geometry in a view window will be removed in direction of the ***Z**-Axis*, in case the drawing was originally created in ***DraftBoard** Unlimited*.

In that way it resembles the **Remove z-coordinates** command in the **Arrange** menu, except that the **Remove z-coordinates** command removes only three-dimensional geometry in direction of the ***Z**-Axis* on the *Sheet* but not in a *Detail View window*.

To flatten all views is recommended when you intend to export a drawing using the export formats **EMF**, **WMF**, **PNG**, **DXF/DWG**

Flattening all Views

1. Select the **Flatten all Views** command the **Window** menu.

   All object geometry visible in view windows will be projected with the current scale factor onto the current sheet.

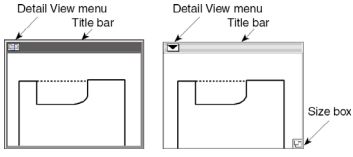
   All overlapping lines of equal length and all lines parallel to the view direction will be removed in view windows.

   The object geometry is no longer associative to the related model.

   Text and hatches will become ordinary object geometry.

View Window

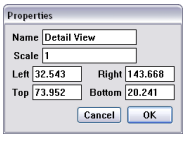
The view window includes a *Title bar*, *Size box* and **pull-down** menu only when the detail view is active.



You can ***drag*** the view around by the title bar and resize the window by ***dragging*** the size box.

The **Detail View** menu provides options for manipulating the view.

Properties The **Properties** option allows you to change the view **scale** and define the locations of its corners.



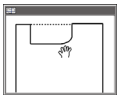
Cut The **Cut** option allows you to remove the view from the sheet and place it on the *Clipboard*. This is useful for placing views on a different sheet. Use the **Paste** command from the **Edit** menu to paste the cut view window.

Copy The **Copy** option allows you to place a copy of the view window on the *Clipboard*. This is useful for pasting multiple identical views onto one or more sheets. Use the **Paste** command from the **Edit** pull-down menu to paste a copy of the view window.

Delete This option deletes the active window and makes the sheet active. It does not delete geometry.

Pan This command allows you to move the geometry inside the view window. A hand icon appears which you can use to ***drag*** the contents of the window.

When you release the mouse button, the **Pan** function ends.



Resize The **Resize** option allows you to ***drag*** a new view window. This function differs from dragging the window borders. With the **Resize** command, you do not have to change each border individually since you define a new window that is substituted for the original one.

Changing the view scale

When you are trying to fit a view or group of views in a drawing format, you may have to scale a view.

1. Choose **Properties** from the **Detail Views** menu.
2. Specify the scale you want and click **OK**.

Zooming within a view

1. Click inside the view to make it active.
2. Hold down the **Shift** key.
3. Choose the **Zoom** command you want from the **Arrange** menu.
   The geometry inside the view is zoomed.

Scrolling within a view

1. Click inside the view to make it active.
2. Hold down the **Shift** key and move the scrollbars.
   The geometry inside the view is moved.

All commands from the **Arrange** pull-down menu work inside the view when you hold down the **Shift** key while choosing from the menu.

All scrollbar functionality can be used inside the view when holding down the **Shift** key while moving the scrollbars.

Bring To Front

This command in the **Views** pull-down menu brings the specified view to the front of the **stack** of views.

Bringing a specified view to the front

1. Select the view.
2. Choose **Bring To Front** from the **Views** menu.

This command counteracts the **Auto Front** command, so when you click, you select objects in the view rather than activating another view.

Send To Back

This command in the **Views** pull-down menu sends the specified view to the back of the **stack** of views.

Bringing a specified view to the back

1. Select the view.
2. Choose **Send To Back** from the **Views** pull-down menu.

This counteracts the **Auto Front** command, so when you click, you activate another view rather than selecting objects.

Auto Front

The **Auto Front** command in the **Views** menu overrides the **Bring To Front** or **Send To Back** settings. When this setting is in effect, you can simply click on a view to bring it to the front and make it active.

Handling overlapping views

The view control commands in the **Views** menu allow you to control the active view. When you have overlapping views, you can think of them as a stack where only the top view can be active. If the **Auto Front** command is turned on and you click a view it comes to the top of the stack and you can work in it. This may cause some problems when you click an object in the top view to select it, but an underlying view is activated instead.

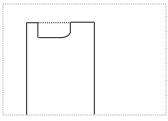
This command only works for views.

To remedy this situation, turn off the **Auto Front** command. Select the view you want to work in and choose **Bring To Front** from the **Views** menu. The objects in the view you brought to the front remain on top, even when you click a different view visible in the top view's space.

Draw View Boundaries

This command in the **Views** menu shows the boundaries of all inactive views. This boundary will be printed if you leave it displayed when you

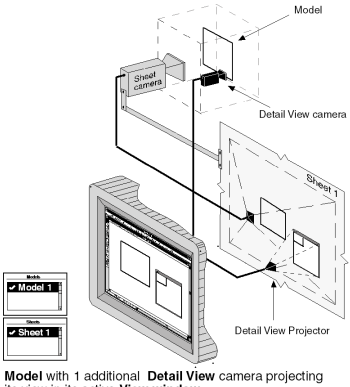
choose the **Print** command. If you want to print the view without boundaries deactivate **Draw View Boundaries** in the **Views** menu.



How Detail Views work

When you create a *Detail View* you don't copy the object geometry (Model) into the *View Window*, but you see the same object geometry (Model) through a second camera—the **Detail View Camera**.

The **Detail View** camera behaves like the **Sheet Camera** except it displays its view of the model in a **View window** on the sheet.



Model with 1 additional **Detail View** camera projecting its view in its active **View window**

All views displaying the same model are associative. For example, when you change the geometry on the sheet, the geometry in the view window changes, and vice versa.

When you have several views displaying the same model they have to be associative, because no matter which view is active—the **Sheet View** or any **Detail View**—you are always editing the same model.

Drawing Formats

When you have completed and scaled your drawing using the **Drawing Size** command, you can place the drawing into a drawing format. For that you can import one of the Standard Drawing formats shipping with ***DraftBoard*** that already contain *border* and *cutting lines* and a *Title block* for entries such as *name* or *scale* of the drawing.

How you create your own drawing formats for your laser printer is described later in this chapter.

You cannot only modify all drawing formats in the **Layout** folder of the **DraftBoard** directory but create as well your own drawing formats and design layouts.

Importing Drawing Formats

1. Scale the drawing using the **Drawing Size** command in the **Layout** menu.
2. Mark the option **Always display Page bounds** in the displayed **Drawing Size** dialog that allows you to compare the true size of the drawing with the displayed printing area of the drawing format.
3. In case your drawing contains any detail views, click with the **Selection tool** on the current sheet outside of any detail view window .

   The window boundaries are displayed by *dotted lines* to indicate that the next step is executed on the current sheet and not within a detail view.
4. Choose the **Import** command of the **File** menu.

   The **Import** dialog box is displayed.
5. Open the folder **Layout** and select the drawing format you want.
6. Click the **Open** button.
7. Mark the option **unscaled** in the displayed dialog box.
8. Click the **OK** button.

The **Drawing size** command displays the printing area of the selected paper size at its true size on the drawing area. You can change the paper format using the **Print Setup** command in the **File** menu.

The *Drawing Format* is imported onto the current sheet. The imported drawing format is *grouped*.

9. Now you can ***drag*** the drawing format on the sheet until all objects lie within the drawing format.

In case the size of the imported drawing format is to small or to large to frame the entire drawing you should delete the drawing format, scale the content of the drawing and import the format again.

Modifying Drawing Formats

All drawing formats provided in the **Layout** folder can be modified to meet your needs. When you want to add new formats is recommended to modify existing ones and save them under a different name.

Drawing formats contain the *border line* of the drawing, the *cutting line* and a *Title block* for entries such as *name* or *scale* of the drawing.

Drawing formats have to be created for each paper format such as *A*, *B*, *C* or *D* and for both paper orientations *Portrait* and *Landscape*.

All drawing formats you find in the **Layout** folder of **DraftBoard** are designed for *Plotter devices*, where the *Cutting lines* correspond exactly with the dimensions of the selected paper format (for example *30 x 40 inch* for the *D* Format) and the *Border line* is offset inside by *1/2 inch* according to the ANSI standard.

When you want to use these plotter formats for your laser printer you have to adjust them to the printing area of your printer, that could differ from printer to printer.

Creating Drawing formats for Laser printers

1. Open a drawing format in the **Layout** folder of the **DraftBoard** folder.
2. Save the drawing under a new name.
3. Select the **Drawing Size** command in the **Layout** menu.
   The **Drawing Size** dialog box is displayed.
4. Mark the option **Display always Page Bounds**.
   The *Printing area* of your active Laser Printer is displayed as a grey rectangle.

***Important**: If the rectangle is displayed vertically you have to change the paper orientation with the **Print Setup** command in the **File** menu and call the **Drawing Size** command again.*

5. Click **OK**.
   The **Drawing Size** command is closed.
6. Mark the grouped drawing format on the drawing area and ungroup with the **Ungroup** command in the **Arrange** menu.
7. Adjust the *border lines* of the drawing format until they are exactly aligned with the displayed grey rectangle for the printing area of your laser printer.
8. Delete all *cutting lines* since they are not required for print outs on a laser printer.
9. Select the complete drawing format using the **Select all** command in the **Edit** menu and group the format using the **Group** command in the **Arrange** menu.
10. Save your work.

CHAPTER

12

DraftBoard Documents

Handling Documents

Data Exchange

DraftBoard Documents

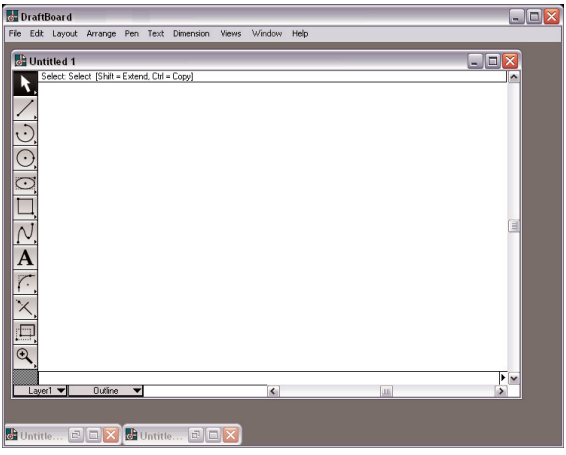
This chapter describes the handling and administration of *Files* (in ***DraftBoard*** called drawings) and the data exchange with other programs.

Using Documents

A document is a ***DraftBoard*** file. Whenever you open ***DraftBoard***, or choose **New** from the **File** menu, a new document appears in its window.

Working with multiple Documents

In ***DraftBoard*** you can open and edit several drawings simultaneously. Each *Drawing* is displayed in its own window that can be reduced to an icon near the lower-left corner of the ***DraftBoard*** work space. To close, move or resize a drawing you use the control buttons in the title bar of each window.

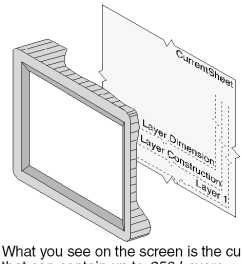


Windows

The **Windows** submenu in the **Views** menu shows the names of all the open ***DraftBoard*** *documents*. To bring a document to the front, that is covered by another document, select its name from the **Windows** submenu.

Components of a Document

A drawing can be a simple part or a complex assembly. Each *Drawing* has one or more *Sheets*, like the sheets of paper in a set of blueprints, and each *Sheet* is made up of *Layers* which can be hidden or displayed as needed.



What you see on the screen is the current sheet, that can contain up to *256 Layers*.

The first part of this chapter discusses using the **File** commands for *opening*, *importing*, *exporting* and *saving* files. Printing and plotting documents is described in the *Basic Drafting* chapter.

Document Properties and Preview

To display *Document Properties* and the *Drawing Previews* in the ***Windows***-*Explorer*, you must select the folder containing the ***DraftBoard*** *Documents*, whose **Attributes** and **Previews** are saved with the drawings, and change the display options for the *Windows Explorer* accordingly.

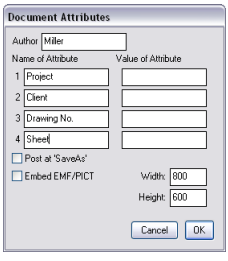
For each drawing you can save specific document properties and a preview of the drawing, that can be optionally displayed in the *Windows* ***Open*** dialog box and in the ***Windows*** *Explorer* .

All these attributes are set with the **Documents** command in the **Preferences** submenu in the **Layout** menu.

Document Properties

This command allows to define document properties and specify if these values should be displayed when saving a drawing. In addition you can specify, if a preview of the drawing should be saved with the document.

When you choose this command the following dialog window is displayed:



The **Document Attributes** dialog window allows the following settings:

Author — In this field you enter the name of the author as creator of the drawing.

Name of Attribute — In these four entry fields you can specify document attributes, that are displayed when you select the **Save as** command. The date of original creation must not be specified, since it will be saved automatically with each drawing.

Value of Attribute — In these fields you enter values for the attributes you have defined in the left column, in case they don't change with each drawing. These values are optional, since they can be entered each time when a drawing is saved.

Post at 'Save as' — When you activate this option, the **Document Attributes** dialog will be displayed automatically when you select the **Save as** command.

Embed EMF/PIC — When you activate this option, a picture of the drawing will be saved automatically with the document each time you save a drawing. The **Height** and **Width** of this picture you can indicate in the respective entry fields. This preview picture will be displayed automatically in the **Open** dialog window and can be displayed in the ***Windows*** *Explorer*.

In order to display document preview and attributes in the ***Windows***-*Explorer*, you must select the directory containing ***Draft-Board***-*drawings* which are saved with preview and attributes. Depending on the *Windows* version the *Explorer* must be configured accordingly.

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Cancel — Clicking the **Cancel** button closes the dialog window and ignores all entries made in the **Document Attributes** dialog box.

OK — When you click this button, the dialog window is closed and all settings are saved for the current drawing.

These setting are only valid for the current ***DraftBoard*** session. When you want to save these settings permanently you have to select the **Save Preferences** command in the **Preferences** submenu in the **Layout** menu.

Defining Document Properties

1. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window is displayed.

2. Specify a name for the author of the drawing in the **Author** entry field.

3. Enter into the **4 Name of Attribute** fields attributes such as *Drawing Number* etc. that should be saved with the drawing. It is not necessary to specify an attribute for the original creation date,

since it will be automatically saved with the document as soon as an entry is made in the **Document Attributes** dialog box.

4. Specify values for the defined attributes, if these values don't change with each drawing. If not, they can be entered each time when you select the **Save as** command and the option **Post at 'Save as'** is activated.
5. Activate the option **Post at 'Save as'**, if you want to display the **Document Attributes** dialog each time you select the **Save as** command.
6. Activate the option **Embed EMF/PIC**, if you want to save a small picture of the drawing as preview with the document, that will be displayed in the **Open** dialog window and optionally in the ***Windows*** *Explorer* (**Win98**, **WinSE**, **WinME**, **Win2000**, **WinXP**).
7. Enter a pixel value for the **Width** and **Height** of the graphic. The default value is **600** x **600**.
8. Click the **OK** button.

   The **Document Attributes** dialog window is closed and the specified values are saved for the current drawing.

   If you want to save these attributes permanently as default values, you have to select the **Save Preferences** command in **Preferences** submenu in the **Layout** menu.

Displaying Document Properties when opening a drawing

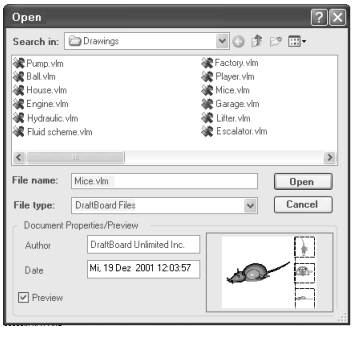
1. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window is displayed.
2. Make all required entries.
3. Click the **OK** button.

   The **Document Attributes** dialog window is closed and the specified values are saved for the current drawing.
4. Save the drawing using the **Save as** command.
5. Close the drawing.
6. Select the **Open** command from the **File** menu.

   The **Open** dialog window is displayed.
7. Select the drawing that you just have saved.

   In the **Open** dialog window a *Preview* of the saved drawing (in case this option was activated for this document), the *Author* (if defined) and the *Original Creation date*. All other defined attributes for the drawing are stored in the **Document Attributes** dialog window after opening the drawing.



Displaying Document Properties in *DraftBoard*

1. Open a drawing that was saved with document properties you have specified in the **Document Attributes** dialog window.
2. Select the **Document Properties** command in the **Preferences** submenu of the **Layout** menu.

   The **Document Attributes** dialog window containing the attributes saved with the **Save as** command is displayed.

Displaying Document Properties in the Windows Explorer

To display *document properties* and *document previews* in the ***Windows-Explorer***, you must go to the directory that contains ***DraftBoard*** *drawings*, that were saved with **Previews** and **Attributes** and activate it. the option **As Web Page** in the **Views** menu of the *Explorers*. Tool-Tips don't work with all Operating systems.

Opening drawings

A detailed description how to open files you find also in the *User Manual* for the *Operating system* of your computer.

The **File** menu contains the **Open** and **New** command to open a drawing. With **New** you create a new drawing while the **Open** command opens an existing drawing.

New Ctrl+N

This command in the **File** menu creates a new ***DraftBoard*** document. The new document has no name (the title bar shows **Untitled 1**), and is set with the default options, such as *pen style* or *grid display*. If you open more than one new document, the subsequent documents are numbered sequentially until you name them by saving.

Open Ctrl+O

Note: *Use the* ***Import*** *command if you want to use drawings of other format types, such as* ***DXF*** *or* ***IGES****, or if you want to bring in* ***ASCII*** *text.*

The **Open** command in the **File** menu opens an existing document, that was created with ***DraftBoard***. This document can be created by current or earlier versions of ***DraftBoard***. The document appears in the drawing area maintaining the same settings as the last time you saved it. If you made any changes to the default settings, such as editing pen styles or hiding layers, those changes remain in effect.

The **Open** dialog box allows you to specify the document you want and lets you change directories or folders if necessary.

Opening a document

1. Choose **Open** from the **File** menu.

   The **Open** dialog box appears.
2. Display in the **Look in** *pop-up* menu the appropriate *Folder* containing the document you want to open.
3. Select in the file type you want to open from the **File Type** *pop-up* menu.

   The list box displays all files of the selected type available in that folder.
4. Click the **File name** you want to open in the list box.

   If you double-click on the file name, the document is opened without clicking the **Open** button.
5. Click the **Open** button.

Recent File List

The number of files contained in the **Recent File List** can be set for *Windows* in the **DraftBoard.ini** file. For more information, see chapter *Setting the Environment*.

Another way to access files that have been opened recently is through the **Recent File List** that appears in the **File** menu after the **Exit** command. This list contains the names and paths of the most recent files that have been opened by ***DraftBoard***.

To open a file from the **Recent File List**, simply select the file from the **File** menu. If the file has been moved since it was last used and the path is no longer accurate, ***DraftBoard*** will ask you to locate the file by providing the standard **Open** dialog box.

Saving a Drawing

You can save a drawing by choosing either **Save** or **Save As** from the **File** menu. The file is stored in the folder you specify.

Save Ctrl+S

Note*: You should save frequently. Even though your drawing appears on the screen, it is not stored on the disk until you save it. Hours of work can be lost because of a power failure. It is also important to save before performing any intricate, multistep manoeuvre. In that way, if the result is not what you had hoped, you can abandon the file by closing it without saving.*

This command in the **File** menu saves the current ***DraftBoard*** *document* to its original folder. If you want to save it to a different folder or with a different name, choose **Save As**.

If you choose the **Save** command and the document has not been saved previously, the **Save As** dialog box appears automatically, allowing you to name the document and specify the folder in which to save it.

If you have named and saved the document before, a brief message appears when you choose **Save**, and the program pauses while it updates the information.

Save As

This command in the **File** menu saves the current document. A dialog box appears so you can name the current document, give it a different name, or save it to a different folder.

Saving a document with a different name

1. Choose the **Save As** command from the **File** menu.
   The **Save as** dialog box appears.
2. If necessary, display a different folder.
3. Type the name you want to use in the **File name** box.
4. Select the type of file you want to save the drawing as in the **Save as type** dialog box.
5. Either press **Return** or click **Save**.

Making a backup

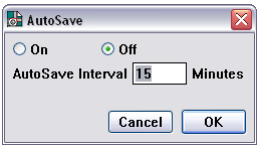
You should make a backup of your work in case you make many changes and want to go back to the original version. You can use the **Save As** command in the **File** menu and save the file with another name or use the **AutoSave** feature described in the next section.

It's also possible that your company makes periodic backups of your folders, and you can go back to a previous version that way.

Auto Save

This command in the **Preferences** submenu of the **Layout** menu directs ***DraftBoard*** to save a backup copy of your work periodically. If your computer hangs up for any reason, the work you did up to the last **Auto Save** will be recoverable.

Selecting the **AutoSave** command displays the following dialog box:



In the **AutoSave** dialog box you can *turn* **AutoSave** *On* or *Off* and specify the *time interval* (in minutes) between *AutoSaves*.

At the specified interval, ***DraftBoard*** creates an *AutoSave file* with the base of the original file name and a **.SAV** extension in the same directory or folder as the original file. **AutoSave** prompts users for a file name *before* creating an *AutoSave file* for untitled documents (created by using the **New** command from the **File** menu). If you supply a file name, this file name will be associated with the document and the *AutoSave file*. If you **Cancel**, **AutoSave** will skip that document, but the prompt will appear again at the next **AutoSave** interval. Files opened with the **READ ONLY** attribute are also *Auto Saved*.

Setting up AutoSave

1. Select **Auto Save** from the **Preferences** submenu of the **Layout** menu.
   The **Auto Save** dialog is displayed.
2. Switch AutoSave **ON** or **Off**.
   The default setting is **Off**.
3. Specify the time interval for *AutoSaving* in minutes.
   The interval can be from **1** to **60** minutes; the default setting is **15** minutes. An interval of **0** or less is equivalent to turning **Auto Save** off.
4. Click **OK**.
   If **AutoSave** is **ON** your work will be saved automatically for the time interval you specified.

If you want to save these settings permanently you have to **Save Preferences** in the **Preferences** submenu of the **Layout** menu.

Closing Documents

DraftBoard provides the **Close** and the **Quit** command in the **File** menu to close documents. If you want to close a drawing without ending ***DraftBoard*** you must use the **Close** command. Selecting the **Quit** command closes all open drawings and ends ***DraftBoard***.

Close Ctrl+F4

This command in the **File** menu closes the current ***DraftBoard*** document (the one displayed in the top window). If other ***DraftBoard*** documents are open, they remain open when you close the current document. If you have made any changes since you last saved the current document, the **Save** dialog box allows you to save the changes. You can close the document with or without saving the changes.

You can also close the document by double clicking the **Control Menu** button at the upper left corner of the title bar.

Quit Ctrl+Q

This command in the **File** menu closes ***DraftBoard***. If you have made changes since you last saved, the **Save** dialog box allows you to save the changes. If you have more than one document open, an alert message allows you to save unsaved documents.

Data Exchange

DraftBoard like any other Computer program saves its data using its own native file format, that can be identified by the file extension **.vlm**. Even though all programs have native file formats that are not compatible to each other, it is possible to transfer data using neutral interchange formats such as **IGES** or **DXF** that standardize the data exchange. When you want to read a file of a non-compatible file format it is called ***Import****ing* files, when you want to save a file in a non-native format type it is called ***Export****ing* files.

It is also possible to import documents of native format type.

Limitations

Even if it is often advertised that programs can directly read and write file formats of competing products, you should be aware that importing files is always limited since the exchange is done through standardized file formats such as **DXF** and **IGES** that can offer only a compromise between the two file formats used.

These filters cannot transfer features specific to one application such as ***DraftBoard's*** *Smart walls* into another application where these features are unknown. Smart walls for example would be exported as a group of simple lines, all smart wall features known in ***DraftBoard*** would be lost.

Importing drawings

The import feature of ***DraftBoard*** allows you to use existing drawings from ***DraftBoard*** or other applications within a ***DraftBoard*** document.

Drawings created with ***DraftBoard*** on the *Macintosh* don't have to be imported but can be read using the **Open** command in the File menu.

The following file types can be imported

DraftBoard ***DraftBoard*** native file format. Displayed option: *DraftBoard Files*. File name extension: ****.vlm***.

TXT **ASCII** text file. Displayed option: *Text Files;* File name extension: ****.txt***.

Spline This option allows to import *Spline coordinates*. ***DraftBoard*** creates then automatically a spline on the drawing area. File name extension: ****.txt***.

WMF **W**indows **M**etafile **F**ormat. An interchange format used by the *Windows clipboard* for graphics commands. Displayed option: *MetaFile;* File name extension: ****.wmf***. *[Windows only]*

BMP Microsoft Windows **Bitmap** file format. The file name extension **.bmp** is automatically appended to the file name. Displayed option: *Bitmap;* File name extension: ****.bmp*** *[Windows only]*.

EPS (AI) ***E****ncapsulated* ***PostScript*** *Format*, a data format for *PostScript* printers or for the import in *PostScript* compatible applications. ***DraftBoard*** supports the ***EPS*** *Format* of the ***Adobe*** *Illustrator* and compatible ***EPS*** *Formats*. Displayed option: *EPS/AI;* File name extension: ****.eps***.

DXF/DWG AutoCAD ***D****ata* e***X****change* ***F****ormat* of ***AutoCAD***. Selecting this format in the **Import** dialog box and clicking the displayed **Options** button, you can choose from the

AutoCAD** Formats* **DXF**, **DWG** *Versions* **10**, **11/12**, **13**, **14** and **2000/2002(2008**. File name extension: *.dwg*** or ****.dxf***.

IGS **IGES 4.0** ***I**nitial **G**raphics **E**xchange **S**pecification*, format to standardize the exchange of graphics. This file format allows as well to import *3D* ***NURB** Surfaces*. Displayed option: *IGES;* File name extension: ****.igs***.

Using the ***DWG/DXF** Import option* you can specify in a ***DXF** configuration file* additional import options such as *color handling, line patterns, line weights* or *text fonts*. More information about this configuration file you find in the *Appendix* of this manual.

Import

This command in the **File** menu imports a document from ***DraftBoard*** or another application into ***DraftBoard*** and places it in the current model. When you select the **Import** command the names of documents ***DraftBoard*** can read are displayed in a dialog box that works like the **Open** dialog box. When you select a document in this dialog box and click the **Open** button, ***DraftBoard*** detects the format of the document and displays the **Import** dialog box:

In the **Import** dialog you can specify the following *Import options*.

Grouped Imports the geometry grouped so it is a single unit in the ***DraftBoard*** document.

Geometry Only Brings in only the geometry, leaving any *text*, *dimensions* and *crosshatching* behind.

Onto Work Layer The imported geometry is placed on the work layer rather than the layer on which it was originally created.

Unscaled The geometry is adjusted to correspond to the scale set by the **Drawing Size** command. If **Unscaled** is specified, the imported geometry is not scaled.

You can ungroup the imported geometry with the **Ungroup** command in the **Arrange** menu.

Importing a document

1. Choose **Import** from the **File** menu.
   A dialog box appears similar to the **Open** dialog box.
2. Select the file type you want to import from the **Files of Type** *pop-up* menu.
   The list box shows all files of the selected type available in that directory.
3. Double-click the **File name** you want to import.
   The **Import** dialog box appears.
4. Specify any import options you want.
   If you have set a scale in drawing size, you should click the **Unscaled** option so that the geometry or drawing format you are bringing in is scaled the same as the geometry in the drawing.
5. Click **OK**.
   The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the **Export** command from the **File** menu.

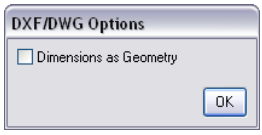
Importing DWG/DXF files

***Important**: When you import a **DWG** or **DXF** file, the geometry is constructed according to the units set in **Preferences** when you choose the **Import** command. Be sure to set the appropriate units before you import **DWG** or **DXF** geometry.*

1. Choose **Import** from the **File** menu.
   A dialog box appears similar to the **Open** dialog box.
2. Select the file type **DXF** from the **Files of Type** *pop-up* menu.
3. Select the ***DXF** file* you want to import and click **OK**.
   The **Import** dialog box is displayed.

4. Select the option **DXF.**
5. Click **OK**.

   The **DXF/DWG Options** dialog box is displayed.



6. Specify the required options.

   When you choose the option **Dimension as geometry**, all dimensions are imported as geometry objects, if not they are imported as dimensions.
7. Click **OK**.

   DraftBoard begins creating the imported file on the drawing area.

! Important !

Important Tips for importing DWG/DXF Files

- When you import a ***DWG*** or ***DXF*** *file*, the geometry is constructed according to the units set in **Preferences** when you choose the **Import** command. Be sure to set the appropriate units used in the ***AutoCAD*** *File* before you import **DWG** or **DXF** geometry.
- When you import a ***DWG*** or ***DXF*** *file*, the **Drawing Scale** in the ***DraftBoard*** **Drawing size** dialog box in the **Layout** menu must be set to **1:1** to make sure that the imported drawing is correctly scaled.
- ***DraftBoard*** supports only **256** layers. If you have a ***DWG*** or ***DXF*** *file* containing more than 256 layers, all geometry on the additional layers will automatically be placed on the current work layer in ***DraftBoard***. If you don't want this to happen, reduce the number of layers to a maximum of **256** in the application you will be creating the ***DWG*** or ***DXF*** *file* in.
- When you get, after importing a ***DWG*** or ***DXF*** *File* into ***DraftBoard***, geometry not contained in the original ***AutoCAD*** *file*, you must purge any unnecessary blocks using the **PURGE** command in ***AutoCAD*** before exporting the file.
- If the file doesn't open correctly you should mark the options **Onto Work Layer** and **Dimensions as Geometry** in the **Import** dialog box and import the file again.

Importing ASCII text from another document

1. Choose **Import** from the **File** menu.

   A dialog box appears similar to the **Open** dialog box.
2. Select the file name you want to import.
3. Click **OK**.

   The **Import** dialog box is displayed.
4. If **Text** is not selected, mark the **Text** option.
5. Click the **OK** button.

   The text appears in a text box in the drawing.

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Importing Splines

When you import a text file that contains the coordinates for a spline ***DraftBoard*** will create the spline according to the imported coordinates.

1. Select **Import** from the **File** menu.

   A dialog box appears similar to the **Open** dialog box.
2. Select a text file that contains the coordinates for the spline.

   The **Import** dialog box is displayed.
3. Specify the import option **Spline**.
4. Click the **OK** button.

   DraftBoard begins creating the Spline.

Creating a text file for importing a spline

1. Use a *text editor*, *word processor* or *spreadsheet* to create a text file.
2. Input **X**, **Y** and **Z** values for your spline coordinates.

The text file should be tab or space separated. Each line ends with a return. Line feeds after each return should have no effect.

The text file should conform to the following columnar format:

1 1 0

2 2 0

You are allowed to specify decimal coordinates as well:

1.33 1.1 0

2.4 2.5 3.5678

Be sure that you do a **Return** after the last coordinate, if not ***DraftBoard*** will not import the coordinates specified in the last line.

Even ***DraftBoard*** is only a 2D Program, the ***Z-Coordinate*** has to be indicated using the value **0**.

3. Save the file as **Text only** and import into ***DraftBoard*** to create the spline.

Projection of 3D Data

When you import a three-dimensional model into ***DraftBoard***, the third dimension is invisible, since the model is displayed in the **Top** view onto the ***X-Y** Plane,* even there exists geometry in direction of the ***Z** axis*.

These unnecessary invisible elements can be projected onto the work plane (x-y) using the **Remove Z-Coordinates** command.

This is always recommended when you export a file to prevent the transfer of disturbing three-dimensional data.

Remove Z Coordinates

This command in the **Arrange** menu projects all elements of a 2D drawing onto the current workplane, that have besides the usual ***X**-, **Y**-Coordinates*, additional ***Z**-Coordinates*.

The **Flatten all Views** command in the **Views** menu works similar, but is valid only for *Detail views*, while the **Remove Z-Coordinates** command does the same for the drawing sheet. See as well under **Flattening all Views** in the *Structuring Drawings* chapter in this manual.

Remove Z-Coordinates

1. Import a three-dimensional model into ***DraftBoard***.

   The body will be displayed in the **Top** view projected onto the ***X-Y**-plane*.

   A circle that is positioned unparallel to any standard plane in a three-dimensional space will be displayed as an ellipse.

2. Select the **Remove Z-Coordinates** command in the **Arrange** menu.

   All objects with a ***Z**-Coordinate* will be projected onto the current workplane.

Exporting a Drawing

All settings for exporting a drawing are specified either automatically or by option buttons in the respective **Export** dialog windows displayed when exporting a file.

Export Data Interfaces

Occasionally, you may need to save a document in a format other than the ***DraftBoard*** format so you can use it with other application software. All settings for the export of drawings are specified either automatically or by option buttons in the respective **Export** dialog windows displayed when exporting a file.

Additional information about exporting data you find in the *Appendix* of this manual.

The following file types can be exported

DraftBoard 4.5 — ***DraftBoard's*** native file format, compatible to ***DraftBoard Unlimited 4.0***.
File extension: ****.vlm***.

Vlm 3.0/4.0X — In ***DraftBoard Unlimited 4.5*** the file format was modified and is therefore incompatible to all previous versions. With the **Vlm 3.0/4.0X** format files can be exported into previous versions. File name extension: ****.vlm***.

Text — **ASCII** characters. This option is available only if in ***DraftBoard*** the **Text** tool is the active tool. File name extension: ****.txt***.

Bitmap — *Microsoft Windows* **Bitmap** file, an interchange format specifically used for pixel graphics. File name extension: ****bmp***. *[Windows only]*

MetaFile — The format is used by the *Windows Clipboard*, an interchange format specifically used for graphics commands. The precision equals the computer screen precision and exports graphics as vectors and pixel as represented on the screen. File name extension: ****.wmf*** for the 16 bit Window version or ****.emf*** for the 32 bit Window version. *[Windows only]*.

PNG — ***Portable Network Graphics-Format***, a graphic format mainly used for distributing Internet graphics. File name extension: ****.png***.

EPS/AI — ***Encapsulated PostScript*** *Format*, a data format for *PostScript* printers or for the export in *PostScript* compatible applications. ***DraftBoard*** supports the ***EPS*** *Format* of the ***Adobe*** *Illustrator* and compatible ***EPS*** *Formats*. File name extension: ****.eps***.

For best possible translations see ***DXF*** *Configuration* in the *Appendix* of this manual.

DXF/DWG — AutoCAD ***Data eXchange Format*** of ***AutoCAD***. This data interface supports the ***AutoCAD*** *Formats* **DXF**, **DWG** of *Versions* **10**, **11/12**, **13**, **14** and **2000/2002**. File name extension: ****.dwg*** or ****.dxf***.

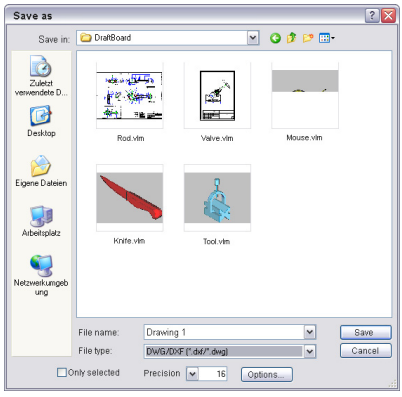
IGES — **IGES 4.0**. ***Initial Graphics Exchange Specification***, format to standardize the exchange of graphics. File name extension: ****igs***.

Export

This command in the **File** menu saves a document in the format you specify.

Exporting a document

1. Choose **Export** from the **File** menu.
   The **Save as** dialog box appears.



2. Enter the desired file name.
3. Select the desired file type.
4. Set the required options.
   Depending on the selected export file type you can set several options, that are described in the next section of this chapter.
5. Click the **Save** button.
   The file is saved with the specified *file name*, *file type* and *export options*.

Export option for the available file types

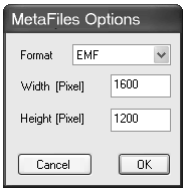
Depending on the selected export format you can set several options:

Only selected — If you wish to export only selected objects on the screen, rather than the entire document, mark the option **Only Selected**.

Precision For some formats you can set the number of decimal places to specify the precision of the exported data.

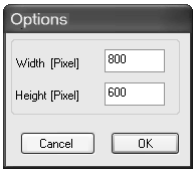
Options The following formats allow to display a dialog box by clicking the **Options** button in which you can set additional export options.

Metafiles The following export option are available:



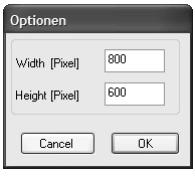
You can choose from the following Metafile Formats: **WMF** (16 bit Windows Version), **EMF** (**E**nhanced **M**etafile **F**ile: 32 bit Windows Version), **EMF** a Aldus specific ***EMF****-Format*. In addition you can specify the **Width** and **Height** of the graphic in Pixel units. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the print out quality.

PNG The following export option are available:



In this dialog window you can specify the **Width** and **Height** of the graphic in Pixel. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the printout quality.

Bitmap The following export option are available:



In this dialog window you can specify the **Width** and **Height** of the graphic in Pixel. These values specify the quality (resolution) of the graphic for the print out. The higher the resolution specified the better the print out quality.

DWG/DXF The following export option are available:

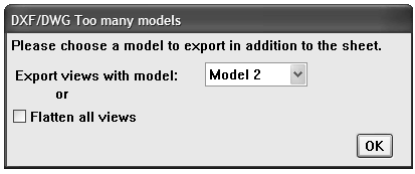
Additional information exporting ***DWG/DXF*** *Data* you find in the Appendix of this manual.

In this dialog window you can select the ***AutoCAD*** *Formats* **DWG**, **DXF**, **DXB** and *Versions* **10**, **11/12**, **13**, **14** and **2000/2002**

For ***AutoCAD*** *Version* ***12***, please choose *Version* ***11***.

In addition you can specify, if you want to export all hatches ungrouped.

Always when on the *current sheet* 2 or more models are displayed in *detail views*, the following dialog box is displayed:



In this dialog window you select either an additional model with its corresponding detail views or flatten all views to export them all.

Important Tips for the Export of DWG/DXF Files

- For exchanging data with other ***CAD** Programs* you should prefer the file format **DXF/DWG 2000** and for exchanging data with ***NC** Programs* the file format **DXF 12**.
- Until *version* ***13 AutoCAD*** doesn't accept any special characters such as a **&** character in layer names. Therefore remove all special characters from layer names before exporting any file into ***AutoCAD***. Otherwise ***AutoCAD*** will not import the file and generate an error message instead.
- In case line patterns are disorted (too big or too small) in ***AutoCAD*** after importing a ***DraftBoard DXF** File*, change the ***AutoCAD*** variable **LTSCALE** to a smaller number to display the line patterns at an appropriate scale.

CHAPTER

13

Symbols

Creating Symbols

Placing Symbols

Symbol Libraries

Symbols

DraftBoard offers powerful functions for creating and handling symbols. All symbol commands are listed in the **Symbols** submenu of the **File** menu.

The following section describes how to *create* and *place symbols* and the configuration of *Symbol Libraries*.

Creating Symbols

In ***DraftBoard*** you can place any drawing as a symbol and open any symbol as drawing.

DraftBoard distinguishes between:

- *Simple Symbols,*
  which have fixed dimensions,
- *Parametric Symbols*,
  which have parametric dimensions and as an option underlying value tables,

  and
- *Symbol Libraries*,
  that are configured with the **Libraries** command in the **Symbols** submenu, that can contain *simple* and *parametric symbols* and are handled exclusively with the **Symbolmanager**.

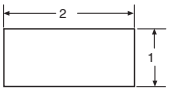
The following sections describe how to create the different type of symbols using a simple rectangle.

Simple or Fixed Symbols

Objects with fixed dimensions are called *simple* or *fixed* symbols.

Creating simple symbols

1. Draw a rectangle with a *width* of **2 inch** and a *height* of **1 inch**.
2. Dimension the rectangle using the **Horizontal** and **Vertical dimension** tools.



When you place a *fixed symbol* in a drawing using one of the **Symbol** commands, it is placed with its dimensions. These dimensions are fixed and can not be modified once the symbol is placed.

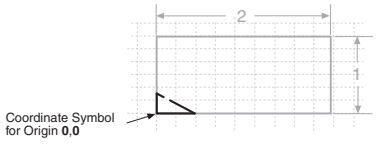
Defining the Insertion point of a Symbol

When you place a symbol in a drawing the *Insertion point* of the symbol is placed at that location you first clicked onto in the drawing area.

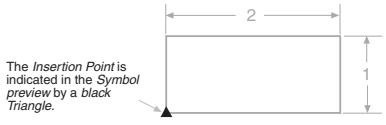
To define the *Insertion point* of a symbol you proceed as follows:

1. Create the object..
2. Place one *control point* at the origin (**0,0,0**) to specify the *Insertion Point* of the symbol.

A new document has its origin (**0,0,0**) in the center of the screen. To see the origin on the screen you must display the *Grid*. If you don't place a *Control Point* into the origin, the most left *Construction Point* is automatically used as *Insertion point*.



In the **Preview** window of all symbol dialog boxes the *Insertion point* of a symbol is shown as a small black triangle.



Parametric Symbols

Parts dimensioned with parametric variables are called *Parametric Symbols*. Parametric symbols always are the first choice whenever parts equal-shaped but with varying dimensions are required.

You cannot create but use *Parametric Symbols* in ***DraftBoard***. Most symbols available in the symbol libraries shipping with ***DraftBoard are*** parametric symbols.

Whenever you place a parametric symbol in a drawing, the symbol is placed without all dimensions containing a parametric variable. Parametric dimensions can be modified after placing a symbol in a drawing in the **Edit Objects** dialog window..

Parametric Symbols with underlying Value Tables

When you place a parametric symbol in a drawing all parametric variables are listed in the **Symbols** dialog box. Whenever a parametric symbol has an underlying value table, you can, instead of entering the final values into the related entry fields, select predefined values using the **Parameter** button in the **Symbol** dialog box.

Value tables for parametric symbols are pure ***Text*** *Files* (***ASCII*** *Files*), that are saved in the **Symbol** folder using the related symbol name with the file extension **.txt**.

Value tables may not only contain parametric variables but also user-defined attributes such as *material* or *order number* that can be extracted in a *Bill of Materials*.

When you place a symbol with an underlying value table using the **Insert** command in the **Symbol** submenu of the **File** menu or the **Symbolmanager**, all defined *Attributes* and *Variables* are listed in the related dialog box and can be modified before placing the symbol in a drawing.

Editing Value Tables

1. Open the value tables with the file extension .txt with a common Texteditor.
2. Edit the table.
3. Save the table.
4. ***Important:*** *Delete in the folder containing the Symbol and the Value Table the file with the related symbol name and the file extension* ***.var.***

Important: *It can take a while when you open a symbol with an underlying value table for the first time since* ***DraftBoard*** *has to convert the ASCII table in an own format with the file extension* ***.var****.*

Placing Symbols

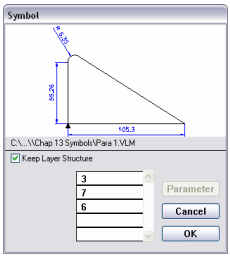
The submenu **Symbols** in the **File** menu contains two commands for placing symbols: **Insert** and **Symbolmanager**.

The **Insert** command allows to place any ***DraftBoard*** drawing and any norm part as a symbol.

Whereas the **Symbolmanager** command allows only to place and edit symbols from *Symbol libraries* that were configured with the **Libraries** command or imported as *Symbol libraries*. (See as well under *Symbol libraries* at the end of this section.)

Insert

Choosing this command from the **Symbols** submenu in the **File** menu displays the following dialog box:



If you want to enlarge part of the displayed symbol, bring the mouse pointer over the area of interest and press the left mouse button. As long as you keep the mouse button pressed the clicked area will be magnified.

The **Symbols** dialog box allows the following settings:

Preview window	The **Preview** window shows a picture of the selected symbol including all its dimensions.
Path	Below the **Preview** window you find the path to the folder where the selected symbol is stored.
Variables	For parametric symbols this list contains entry fields for all variables defined for the symbol.
Parameter	When you click the **Parameter** button the **Parameter** dialog box is displayed containing all attributes defined in the corresponding value table. If no value table exist the **Parameter** button is deactivated.

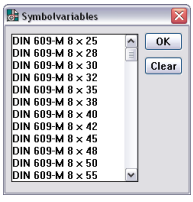
To place a symbol in your drawing click or ***drag*** a vector on the drawing area. If you click, the symbol is inserted in the location you click, in its original orientation. If you ***drag***, the starting point of the ***dragged*** vector specifies the *Insertion point* for the symbol and the direction of the ***dragged*** vector indicates the *orientation*.

When you click **OK** the symbol is placed into the drawing according to the selected *Parameters* and the defined *Insertion point* and *Orientation* and at the current scale of the drawing.

Placing a symbol with the Insert Command

1. Choose the **Insert** command from the **Symbols** submenu.
   The **File open** dialog box is displayed.
2. Select the wanted symbol file and click the **Open** button.
3. Enter a value for each parametric dimension or click the **Parameter** button, if an underlying value table for the symbol exists.
   The **Parameter** dialog box is displayed.

The **Parameter** button is only active when a underlying value table exists.



4. Select a symbol from the **Parameter** list.
   The symbol values are transferred into the related entry fields of the **Symbol** dialog box.
5. Click the **OK** button.
   The **Parameter** dialog box is closed.
6. Edit the symbol values, if desired.
7. ***Drag*** a vector on the drawing area for the *insertion point* and the *Orientation* of the symbol.
   In the **Symbol** dialog box a *black triangle* indicates the *Insertion Point* of the symbol.

If no *Insertion Point* was specified when the symbol was created, the most left *construction point* of the symbol will be automatically used as insertion point.

8. Click **OK**.
   The geometry resolves and appears in the current drawing at the location you clicked and sized as you specified. The inserted symbol is grouped and selected.

Placing a symbol with its parametric dimension

If you want to bring a symbol including its parametric dimensions into the current document you must use the **Import** command in the **File** menu. The symbol is placed at its original size including all its dimensions into the current document.

Standard, *non-parametric* dimensions are always placed with the symbol and must not be imported.

Editing parametric Symbols

If you want to edit parametric symbols you must select the symbol and choose then the **Edit Objects** command in the **Edit** menu.

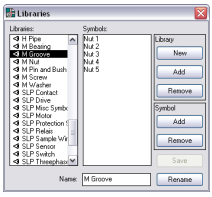
Symbol Libraries

In addition to the symbol libraries shipping with the ***DraftBoard*** you can create your own libraries using the **Libraries** command from the **Symbols** submenu in the **File** menu.

Libraries

This command allows to configure symbol libraries containing *simple* and *parametric* symbols.

The **Libraries** command displays the following dialog box:



The **Libraries** dialog box allows the following settings:

Libraries — This list field contains all available symbol libraries.

Symbols — This list field shows all symbols of the selected library.

New — This button creates a new library. When you click the **New** button the **Set Library file** dialog box is displayed where you can select a folder for the new *Library reference file* and overwrite the default library name **Library.vlb**. When you click the **Save** button the dialog box is closed and the library list shows the new library with the default name **Library 1** that you can modify in the Rename field.

Add — *Library*
This button allows to add an existing library. When you click the **Add** button the **Add Library** dialog box is displayed where you can select a *Library reference file* with the file extension **.vlb**. When you click the **Open** button the dialog box is closed and the selected library is added to the existing libraries.

Symbol
This button allows to add a symbol to the selected library. When you click the **Add** button the **Add Symbol** dialog box is displayed where you can select a symbol file with the file extension **.vlm**. When you click the **Open** button the dialog box is closed and the selected symbol is added to the existing library symbols.

Remove — This button removes a selected symbol or library from related list field.

Rename — This button allows to modify the name of a selected symbol or library.

Save — This button saves all changes made in the **Libraries** dialog box. When you close the dialog box using the **Close** field in the title bar, a message reminds you that all changes will be lost without saving.

Creating a new Symbol library

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Click the **New** button.
   The **Set Libraries File** dialog box is displayed.
3. Overwrite the default name **Library.vlb** with an appropriate name.
4. Select a folder where you want to save the *Library reference file*.

The name for the *Library reference file* must not be identical with the name you choose for the library in the **Library** dialog box. It is recommended to save the *Library reference file* in the **DraftBoard** folder.

5. Click **OK**.
   The dialog box is closed and the new library is added to the existing symbol libraries with the default name **Library 1**.
6. Select **Library 1**.
   Library 1 is displayed in the **Rename** field.
7. Overwrite **Library 1** with an appropriate name.
8. Click the **Rename** button.
   The new library name is displayed in the **Library list** field.

Adding Symbols

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Select the library you want add symbols to.
3. Click the **Add** button in the *Symbols* section.
   The **Add Symbols** dialog box is displayed.
4. Choose a *Symbol file* you want to assign to the selected library.
5. Click the **Open** button.
   The dialog box is closed and the new symbol is added to the **Symbol** list of the selected library.
6. Optional you may modify the file name using the **Rename** button.

*Important: When you assign a symbol to a library the symbol will not be copied but only the path to the folder where the symbol is stored. This path is saved in the **Library reference file** of the selected library. When you move a symbol to another folder the saved path becomes invalid and the symbol cannot be found anymore by the library.*

Adding Libraries

1. Choose the **Library** command from the **Symbols** submenu.
   The **Libraries** dialog box is displayed.
2. Click the **Add** button in the *Library* section.
   The **Add Library** dialog box is displayed.
3. Select a *Library Reference file* with the file extension **.vlb**.
4. Click the **Open** button.
   The dialog box is closed and the new library with all its symbols is added to the existing libraries.
5. Using the **Rename** button you can modify the file name of the new library.

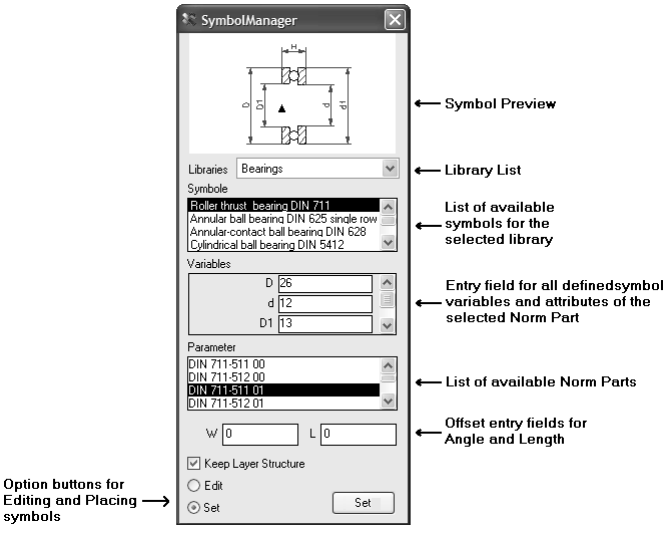
Handling of Symbol libraries

The *Symbolmanager* is the perfect tool for placing and editing symbols. Since the *Symbolmanager* handles exclusively symbol libraries it is recommended to group often used symbols into symbol libraries.

When the *Symbolmanager* is displayed you can place any number of symbols, replace and edit *Norm Parts* and offset the symbol *insertion point* by a specified distance and angle.

Symbolmanager

When you choose this command from the **Symbols** submenu in the **File** menu the **Symbolmanager** is displayed.



The **Symbolmanager** allows the following settings:

Preview window	The **Preview** window displays a picture of the selected symbol.
Library list	This list field displays all available libraries.
Symbols	This list field displays all symbols of the selected library.
Parameter	This list field shows all *Norm parts* of the selected symbol if an underlying value table exists.
Variables	This list field displays all *variables* of the *Norm part* selected in the **Parameter** list. All values of these variables can be modified.
A (Angle)	When you want to place a symbol offset to the first location you click on the drawing area you must specify in this entry field an **Angle** for the offset direction.
L (Length)	When you want to place a symbol offset to the first location you click on the drawing area you must specify in this entry field an offset **Distance**.
Keep Layer Structure	When you activate this option the original *Layer Structure* of a grouped symbol will be kept, when you ungroup it using the **Ungroup** command in the **Arrange** menu.
Edit	When you mark this option the **Set** button will show **Apply** and the mouse pointer changes its shape into a wrench tool. When you select a symbol on the drawing area all related data of the selected *Norm Part* are displayed in the *Symbolmanager*. You can modify the symbol data or select another *Norm Part*. When you click the **Apply** button the symbol will be reshaped or substituted.
Set	This option allows you to place symbols on the drawing area. To place a symbol you must ***drag*** a vector on the drawing area for the *insertion point* and the *orientation* of the symbol. When you click the **Set** button the symbol is placed according to the *dragged* vector. When you have specified an offset angle and distance the symbol is placed offset to the insertion point according to the specified values.

Placing Library symbols

1. Display the *Symbolmanager* using the **Symbolmanager** command from the **Symbols** submenu in the **File** menu.
2. Select a library in the **Library** *pop-up* list field.
   The **Symbols** list field displays all symbols available in the selected library.
3. Select a symbol in the **Symbols** list field.
   If the symbol has an underlying value table all variations of this symbol are listed in the **Parameter** list field.
4. Select a *Norm Part* in the **Parameter** list field.
   All parametric *Variables* and *Attributes* of the selected *Norm Part* are displayed in the **Variables** list field.
5. If necessary you can modify the predefined values.
6. Mark the **Set** option.
7. If you want to place the symbol *offset* from the clicked *insertion point* specify an **Angle** and a **Distance** in the related entry fields.
8. ***Drag*** a vector on the drawing area for the *insertion point* and the *orientation* of the symbol.
9. Click the **Set** button.
   The symbol is placed according to the *dragged* vector.

10. Place additional symbols.
11. Close the *Symbolmanager* using the **Close** box in the upper right corner of the title bar.

Editing Library Symbols

1. Display the *Symbolmanager* using the **Symbolmanager** command from the **Symbols** submenu in the **File** menu.
2. Mark the **Edit** option.
   The mouse pointer changes its shape to a wrench tool when you bring it over the drawing area.
3. Select the symbol on the drawing area you want to edit.
   All related data of the selected *Norm Part* including its symbol library are displayed in the *Symbolmanager*.
4. Modify the symbol data or select another *Norm Part* in the **Parameter** list field.
5. Click the **Apply** button. .
   The symbol will be reshaped or substituted. When you have specified any offset values the symbol is placed according these values on the drawing area.
6. Edit additional symbols.
7. Close the *Symbolmanager* using the **Close** box in the upper right corner of the title bar.

All symbol variables and attributes can be also modified using the **Edit objects** command.

***Note**: Depending on the symbol it can happen that after modifying a variable value it becomes impossible to resolve the symbol again.*

CHAPTER

14

Geometric Analyses

2D Analysis

Geometric Analyses

After you have constructed your geometry, you may want to do some analysis for either engineering purposes or for the production of your design part. ***DraftBoard*** includes a *2D Analysis* feature as well as a *Bill of Materials* feature. This chapter is divided into two sections to discuss these areas.

2D Analysis

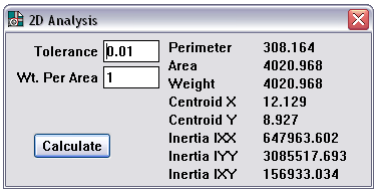
The **2D Analysis** provides sectional properties for the selected geometry, a feature that is useful for many calculations related to design and drafting.

2D Analysis

This command in the **Layout** menu displays the statistics on the selection—the length of the *perimeter*, the enclosed *area*, the center of gravity (centroid), and moments of *inertia*. The selection must be a closed figure.

It is also possible to determine the length of not enclosed objects such as *Splines* or *Arcs*. (See as well under *Perimeter* in this section of the manual.)

When you select the **2D Analysis** command the following dialog box is displayed:



The **2D Analysis** dialog box allows the following settings:

Tolerance A setting to determine the accuracy of calculations for curved objects. The smaller the tolerance is, the higher the accuracy will be. The default value is **0.01**. You can change the value by entering a new number.

Note that this *Tolerance* value has no effect on the precision of the resultant values. The precision of the result is set using the **Units** command under **Preferences** in the **Layout** menu.

The 2D Analysis mechanism analyses *arcs*, *circles*, *ellipses*, and *splines* as straight line segments that deviate from the true curve by no more than the stated tolerance value. The smaller the number, the greater the accuracy; however, the greater the accuracy, the longer the calculation time.

Weight Per Area A multiplier used to determine the actual weight of a part of constant thickness defined by a selected boundary. The Weight of the part is the product of the **Area** multiplied by the **Weight Per Area**.

The default entry is **1**. You can change the value by entering a new number. You can find the proper number for this entry in a handbook published by most material vendors, particularly for sheet steel and aluminium. Enter the weight for the thickness of the proposed material and click the **Calculate** button.

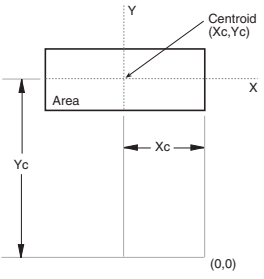
Perimeter The length of all segments which define the selected boundary. This is the only value that is calculated if the boundary of an object (such as a *Spline* or an Arc) is not closed.

For *circles*, *arcs*, and other curves, the accuracy is dependent upon the *tolerance setting*.

Area The surface area enclosed by the selected boundary.

For circles, arc, and other curves, the accuracy is dependent upon the tolerance setting.

Weight The product of the **Area** multiplied by the **Weight Per Area**.



Centroid X The **X** coordinate of the center of mass defined by the selected boundary.

Centroid Y The **Y** coordinate of the center of mass defined by the selected boundary.

Inertia IXX The moment of inertia about a central X-X axis which is parallel to the X-axis.

$$IXX = \int (y-Yc)2dA$$

Inertia IYY The moment of inertia about a central Y-Y axis which is parallel to the Y-axis.

$$IYY = \int (x-Xc)2dA$$

Inertia IXY The moment of inertia about a centroid.

$$IXY = \int (x-Xc)(y-Yc)dA$$

Performing 2D Analysis

The closed figure is an area that could be crosshatched. If you are in doubt about the area, simply crosshatch the geometry to see if it represents the area you want to analyse.
Once you crosshatch the area and you see that your selection is correct, you can choose **Undo** to remove the crosshatching and still maintain the selection.

1. Select the geometry that defines a closed boundary. You may want to use the **Tracer** tool to select the perimeter.
2. Choose the **Construction** command from the **Layout** menu.

   The **Construction** dialog box is displayed, the analysis is performed and the values are displayed in the dialog box.
3. If necessary, enter changes for the **Tolerance** and **Weight per Area** values.
4. Click the **Calculate** button.

   The analysis is recalculated and the new values are displayed in the dialog box.

Calculations

DraftBoard calculates the values for **2D Analysis** before the dialog box appears. If you make a change in the **Tolerance** or the **Weight Per Area**, you must click the **Calculate** button to recalculate the statistics.

The **2D Analysis** mechanism evaluates the boundary in the same way the crosshatching mechanism does; for example, ***DraftBoard*** considers a circle inside a bounded area to be a hole. Crosshatching does not fill the hole and **2D Analysis** does not include the area of the circle in the area calculation.

Displaying the centroid

1. Select the geometry defining the part.
2. Choose the **2D Analysis** command from the **Layout** manu.

   The **2D Analysis** dialog box is displayed.

The **2D-Analysis** dialog box remains open.

3. Choose the **Construction** command from the **Layout** menu.

   The **Construction** dialog box is displayed.
4. In the **Angle** box in the Construction dialog box, enter **0;90**. (Be certain to separate the numbers with a semicolon.)
5. Enter the value for **Centroid X** displayed in the **2D Analysis** dialog box into the **X** box of the **Construction** dialog box.
6. Enter the value for **Centroid Y** displayed in the **2D Analysis** dialog box into the **Y** box of the **Construction** dialog box.
7. Press the **Enter** key.

   The intersection of the two displayed construction lines show the centroid of the selected part.

CHAPTER

15

Basics

Starting ***DraftBoard***

Drawing a Part

Stroke Commands

Fillet and Chamfer

Adding the Holes

Making Changes

Dimensions

Crosshatching

Stretching

Rotating

Printing

Basics

This tutorial is primarily for engineers, designers, drafters, architects, and technical illustrators, especially those who are new to computers. The exercises demonstrate ***DraftBoard's*** most powerful features so you can see how the patented *Drawing Assistant* simplifies **c**omputer-**a**ided **d**esign and **d**rafting (CADD).

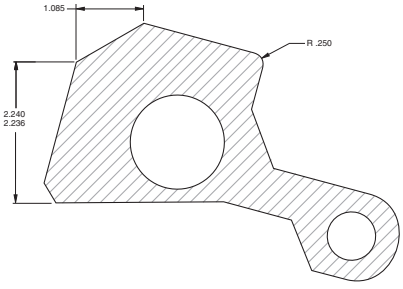
You will soon integrate what you already know about design and drafting with the power of a computer, making your work faster, more accurate, and more creative!

If you are new to computers, CAD may feel awkward and uncomfortable at first. A frustrating truth about computers is that you have to figure out how the computer wants it done. However, once you see what you can do and how easy it is to make changes such as adapting a standard part to new specs, we think you'll agree that the "kick in the side of the head" is worth the effort.

So, be prepared to look at things differently. Go through these exercises, and then begin to adapt DraftBoard's power to your personal work style.

What You Will Construct

In these exercises you will construct this part using computerized *fillets*, *chamfers*, *crosshatching*, and *dimensions*.



You will perform the following tasks:

- Open ***DraftBoard***
- Choose from menus
- Create geometry
- Change geometry
- Use the *Drawing Assistant*
- Create construction lines
- Save a document
- Stroke to create construction lines
- Stroke to zoom
- Create chamfers and fillets
- Construct circles
- Trim
- Change the characteristics of lines
- Dimension
- Crosshatch
- Stretch
- Rotate
- Print

Conventions

The numbered tasks in these exercises describe the activity you are to perform, and the bulleted steps beneath the numbers tell you how to accomplish the task. If you already know how to accomplish the task, you should do it without following the bulleted directions, and then proceed to the next numbered task.

Notes in the margin provide extra information throughout the exercises. You can omit reading these notes and still perform the tasks described. If you want to learn more about ***DraftBoard***, however, you should read them as you go or come back to them once you finish the exercise.

Exploring

Some users like to go off on their own to explore while going through the exercises of a tutorial. This is an excellent way to learn more about ***DraftBoard***. If you are adventuresome, open a new document for your explorations and then switch back to the tutorial document when you want to continue with the exercises.

Occasionally, the tutorial may verify a position or entry that seems obvious to you. If the condition is vital to the next step and you might have inadvertently deviated from the tutorial path, verification (for example, "the **x**,**y** location is **0,0**") has been added to ensure that you get the correct result from the exercise.

There's More than One Way

DraftBoard often provides more than one way to perform a task. This tutorial describes only one method at a time and may show you a different way to do the same task later. When you start to develop a preference, feel free to substitute your own method for whatever is suggested here, provided you're certain that your method produces the same outcome as the tutorial.

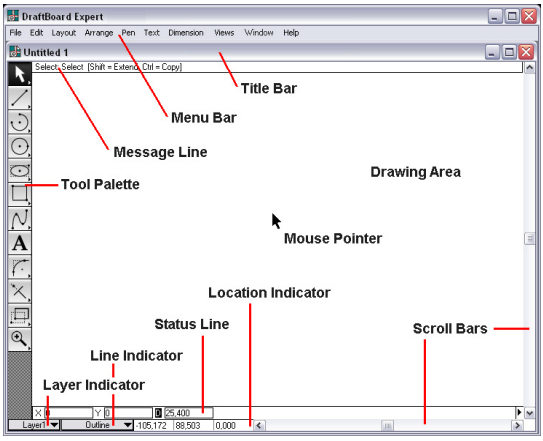
1

Starting DraftBoard

In this exercise, you will start ***DraftBoard***.

1. Start your Computer.
2. Choose **DraftBoard** in the **DraftBoard** submenu of the **Programs** menu from the *Windows* **Start** menu.

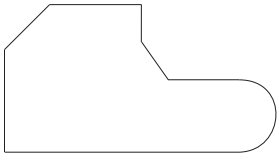
DraftBoard opens, displaying a document named **Untitled 1** with an empty drawing area.



2

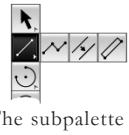
Drawing a Part

In this exercise, you will construct the basic shape of the part.



Since all dimensions in the following exercises are given in **centimeters**, you must select the unit **centimeters** with the **Units** command from the **Preferences** submenu in the **Layout** menu.

1. Select the **Connected Lines** tool.
   - Move the pointer to the **Single Line** tool icon on the tool palette and press (hold down the left mouse button) the **Single Line** tool.

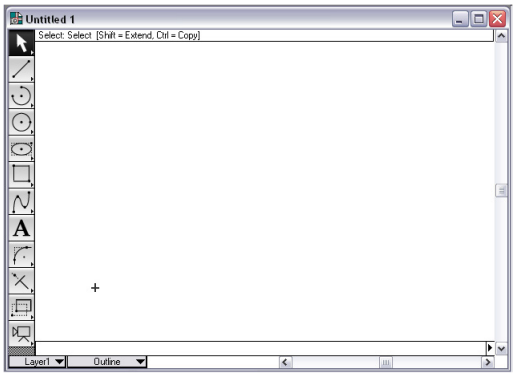
     The subpalette appears.

   - ***Drag*** across the subpalette until the **Connected Lines** tool is highlighted, then release the mouse button. ***Drag*** means to press the mouse button, move the mouse, then release the mouse button.

     The message line tells you that the **Connected Lines** tool is in effect. It remains the active tool until you choose another tool.

Note that the icon on the left of the subpalette is part of the tool palette display and the other icons on the subpalette represent the choices you can make. The icon on the palette changes to reflect your last tool choice, but the icons on the subpalettes are always in the same order, regardless of which tool is on the palette.

2. Draw the first vertical line **8** cm long.
   - Position the pointer in the lower left of the drawing area approximately as shown by the **x** below.



- Click to set the first point.
- Move the pointer up until the *Drawing Assistant's* vertical construction line appears.

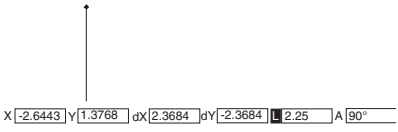
The *Drawing Assistant's* construction lines aid in precise placement. The predefined construction lines are **vertical**, **horizontal**, and at a **45°** angle to existing geometry points. When the pointer is near such a location, the construction line appears and the word ***on*** appears next to the pointer. The *Drawing Assistant* doesn't require you to be exact, only close enough to display the feedback. Once you click, the *Drawing Assistant* locks onto the exact location.

Whatever you type will automatically go into the **Length** status box.

- Move the pointer a couple of inches from your last point (you don't need to be exact), and when ***on*** appears on the construction line, click to set the new point on the construction line.

  At the bottom of the ***DraftBoard*** *window*, the status line displays these boxes.

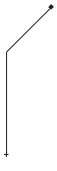
- The numbers on your screen may not match the numbers in this illustration.
- Use either the keypad or the numbers at the top of the main keyboard to type **8** and press **Enter**.



  The **length** goes into the status box and the line is redrawn to exactly *8 cm*, beginning at the first position you indicated.

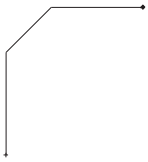
3. Construct the second line **4** cm long at a **45°** angle at the end of the first line. (Angles are measured from horizontal, not from the previous line.)

- Click to set a new point *3 cm* or so from your last point.
- Type **4** and press **Enter**.



  The line segment is constructed at a **45°** angle.

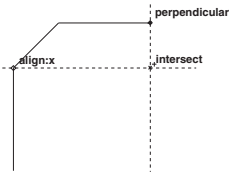
4. Construct a horizontal line **6 cm** long.
   - Move the pointer to the right to display ***on*** for the horizontal construction line.
   - Click a few centimeters from your last point.
   - Type **6** and press **Enter**.



A *6 cm* horizontal line is drawn.

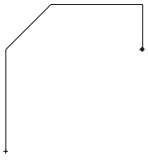
5. Construct a vertical line that ends on the horizontal construction line through the upper ***endpoint*** of your first line.

   - Move the pointer straight down until ***intersect*** appears, then click to set a point at this intersection.



Whenever you move the pointer around the screen, the *Drawing Assistant* examines the existing geometry and displays relevant information about your current position.

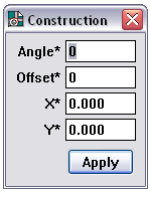
   A vertical line is drawn to the *Drawing Assistant's* ***intersection*** marker.



6. Use a command from a menu to create a construction line beginning at the endpoint of the last line and at an angle of **-55°** and then use it to draw a **3-cm** line.

   - Move the pointer to the word **Layout** in the menu bar.
   - Press and hold down the mouse button and ***drag*** down the **Layout** menu until **Construction** is highlighted.
   - Release the mouse button.

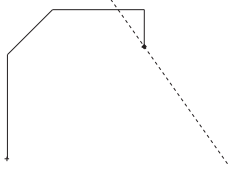
The current tool remains in effect while you choose commands from the menu.



The **x**,**y** coordinates displayed represent the location of the last position you selected, which is the location you need for this step. You can change the coordinates whenever you want.

   The **Construction** dialog box appears. When the pointer is in the dialog box, the angle box is highlighted. You can enter an **angle** for the construction line or an **offset** from the point specified by the coordinates. In this step you are entering an angle for the construction line.

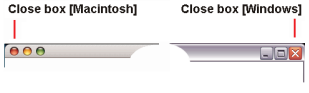
   - Enter **-55** (be sure to type the minus sign).
   - Click **Apply**.



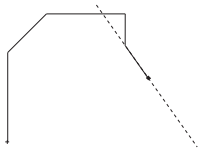
Construction lines created in this way are removed when you choose **Delete Constructions** from the **Layout** menu. Such construction lines are treated like any other line by the *Drawing Assistant*, except that they are placed on the **Construction** layer.

   A permanent construction line appears as a dotted line on the screen.

   - Close the **Construction** dialog box by double-clicking the **Close box** on the upper-right corner.



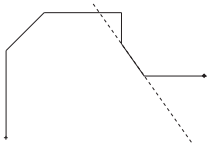
   - Click ***on*** the construction line, about *2 cm* from your last point.
   - Type **3** and press **Enter**.



The line is constructed at a **-55°** angle.

7. Construct a **6 cm** horizontal line.
   - Move the pointer to the right to display the horizontal construction line through the ***endpoint*** of the last line.
   - Click.
   - Type **(5+7)/2** in the **length** status box and press **Enter**.

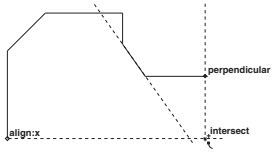
The status boxes accept mathematical operators so that you can add, subtract, multiply, divide, and group values within parentheses. See *Appendix A* for a complete list of mathematical operators.



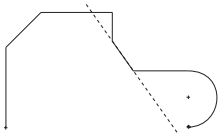
The *6 cm* line appears.

8. Construct an arc from the right ***endpoint*** of the last line to a point horizontally aligned with the first point of this exercise.
   - If part of the construction is now off-screen, choose **Zoom Out** from the **Arrange** menu to display the entire part on the screen.
   - Hold down the **Ctrl** key, then move the pointer straight down until **intersect** appears. (***Note****: The pointer changes to an arc.*)

The message line tells you how you can modify the behaviour of ***DraftBoard's*** drawing tools. For example, if you hold down the **Ctrl** key while using the **Connected Lines** tool, you draw an arc instead of a straight line. The pointer changes into the shape of an arc to signify this optional drawing mode.

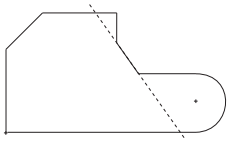


   - Click the mouse button.
   - Release the **Ctrl** key.



The arc appears.

9. Close the figure.
   - Move the pointer horizontally to the left to display the ***endpoint*** notation for the first line you created.
   - Double-click to set this point and signal that the connected-line figure is complete.



The outline of the part is now complete.

10. Save the drawing.
    - Choose **Save** from the **File** menu.

      A dialog box appears so that you can name the drawing.
    - Type **part1** and press the **Enter** key.

It is important to save often. Power failures and mistakes can destroy hours of work if you don't save frequently. It is also important to save before you try any new multistep operation so that you can revert to the original if you don't like the results.

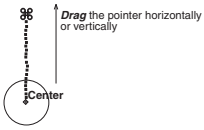
Accomplishments

- Selecting a tool from the tool palette
- Constructing a part with the **Connected Lines** tool
- Using the *Drawing Assistant*.
- Choosing from a menu

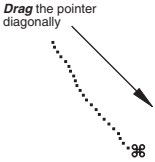
- Creating a construction line
- Saving a document

Stroke Commands

Before proceeding with the drawing, examine the use of ***DraftBoard's*** stroke commands for creating construction lines, zooming, and displaying points. For stroke commands, hold down the **Ctrl** and **Shift** key and ***drag*** the pointer across the screen or click as needed.



Drag	**Construction Lines**
Vertically	A vertical construction line through the first point of the stroke.
Horizontally	A horizontal construction line through the first point of the stroke.

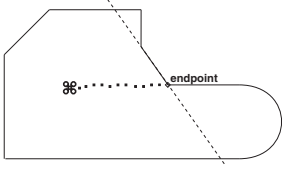


Drag **diagonally**	**Zoom**
Upper left to lower right	Zoom-in enlargement centered over the stroked area.
Lower right to upper left	Reverses zoom-in stroke to the previous magnification.
Upper right to lower left	Zoom-out reduction, the current screen reduces to the size of the area defined by the stroke.
Lower left to upper right	Reverses zoom-out stroke to previous magnification.
Click	**Point Display**
On object	The display of the object's points is turned **on** or **off**.

This command corresponds with the **Show/Hide Points** in the **Layout** menu.

You can use stroke commands while you are using any tool. The possible effects you can obtain with a stroke command depend on the direction you move the pointer. Even though you can create construction lines and zoom in other ways, the stroke commands come in handy because you can use them while you are in the process of using tools from the palette.

1. Create a horizontal construction line.
   - Hold down the **Shift** and **Ctrl** key.
   - Position the pointer at the lower end of the **-55°** line so that ***endpoint*** or ***intersect*** appears.
   - ***Drag*** left horizontally.

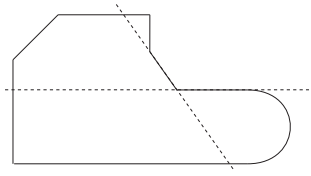


These construction lines are placed on the construction layer and can be removed by the **Delete Constructions** command on the **Layout** menu. Since the work layer is transparent, you can see everything on the construction layer.

The pointer trails a dotted line as you ***drag***. Don't worry if the

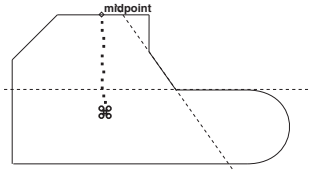
line isn't straight.

- Release the mouse button.



A horizontal construction line appears. Now, you have two construction lines which remain until you delete them.

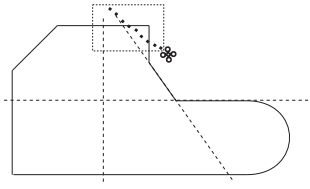
2. Add a vertical construction line.
   - Hold down the **Shift** and **Ctrl** key.
   - Position the pointer at the ***midpoint*** of the uppermost horizontal line.



   - ***Drag*** down vertically.

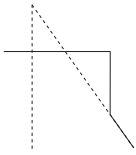
     A vertical construction line appears. You'll use these two construction lines later in the tutorial.

3. Use the stroke command to zoom in on a corner of the drawing so that you can fillet it in the next exercise.
   - Hold down the **Shift** and **Ctrl** key.
   - ***Drag*** as illustrated from above and to the left of the upper-right corner of the drawing across the corner.



The corner is magnified.

Since all tools work at any magnification, you can repeat this action for further magnification.



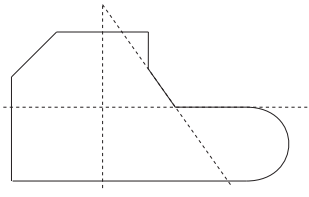
Accomplishments

- Creating a construction line by using a stroke command
- Zooming in with a stroke command

4

Fillet and Chamfer

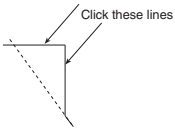
In this exercise, you will fillet one corner of the part and add a chamfer to another.



1. Fillet the corner you zoomed in on in the last exercise.

- Click the **2-Entity Fillet** on the tool palette.

- Click the horizontal and vertical lines that intersect to form the corner.



If the geometry is large, you can hold down the **Shift** key and click inside the corner for single-click filleting. You will use this technique when you create the chamfer in *Step* ***3***.

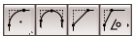
The fillet is drawn with a radius of *0,8 cm*, and the corner is automatically trimmed away.

Fillets are automatically constructed with a **0,8** radius, but you can change the radius in the status line, just as you changed the line **length** earlier. If you don't want to trim a fillet you can hold down the **Ctrl** key while you click the lines that you do not want trimmed away.

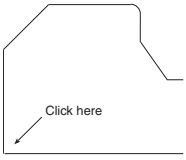
2. Return the drawing to its original size.
   - Hold down the **Shift** and **Ctrl** key.
   - ***Drag*** the pointer from the lower right to the upper left of the drawing area.
   - Release the **Shift** and **Ctrl** key.

It doesn't matter where you start this stroke or how far you ***drag*** if you only want to restore the original magnification.

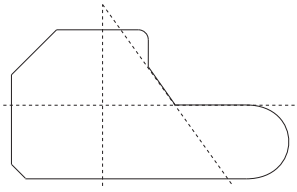
3. Add a chamfer to the lower-left corner of the part.
   - Press the **Fillet** tool to display the subpalette.
   - Select the **Chamfer** tool.



   - Hold down the **Shift** key.
   - Click inside the lower-left corner of the part.



   The chamfer is drawn *0,8 cm* from the original corner.



4. Save your work.
   - Choose **Save** from the **File** menu.

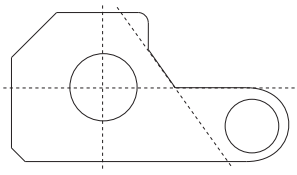
   Since the drawing is already named, the part is saved without displaying the dialog box. The length of time it takes to save depends on the complexity of the part you are drawing.

Accomplishments

- Constructing a fillet
- Zooming out by using a stroke command
- Constructing a simple chamfer

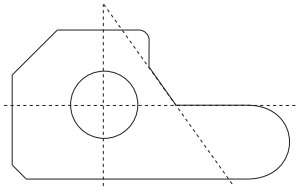
Adding the Holes

In this exercise, you will add two holes to the part, one of which will be offset from a specified location.

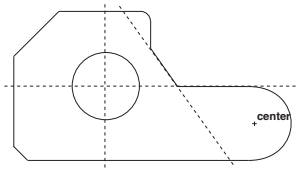


1. Construct a hole, **5** cm in diameter, centered at the intersection of the vertical and horizontal construction lines you created in *Exercise **2***.
   - Click the **Center-Point Circle** tool.

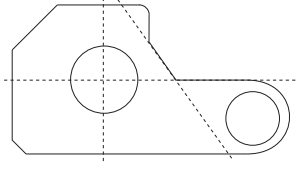
   - Click the ***intersection*** of the horizontal and vertical construction lines to indicate the center of the circle.
   - Move the pointer an inch or so in any direction and click. A circle is drawn with the center point in the location you specified.
   - Enter **5** in the **D** (Diameter) box on the status line.
   - Press **Enter** key.

   The circle is redrawn with a diameter of **5** cm.
2. Construct a **4-cm** hole that is offset **0,2** cm in the negative **x-**direction from the center of the arc of the part.
   - Move the pointer over the arc until the *Drawing Assistant* displays the **+** indicating the ***center*** of the arc.
   - Move the pointer to the **+** to display the ***center*** feedback.

   The **Center-Point Circle** is still the current tool.
   - Click to position the center of the circle.
   - Enter **4** in the **D** (Diameter) box but do not press **Enter**.
   - Click in the **X** box on the status line to place the text cursor at the end of the existing entry.
   - Type **-0,2** (don't forget the minus, you are creating an equation from the existing **X** value) and press **Enter**.

   The circle is drawn with a diameter of **4** cm and its center is offset by **-0,2** cm.
3. Save your work.

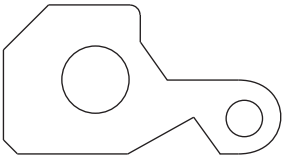
You see here that you don't actually have to draw the circle, just click the center and enter a value for the diameter.

Accomplishments

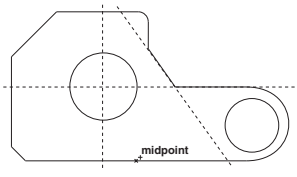
- Constructing circles
- Using point offsets

Making Changes

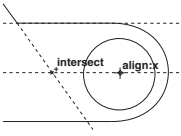
In this exercise you will modify the part you've been constructing. First, you'll add a cutout section to the bottom edge, and then you'll change the diameter of one of the holes.



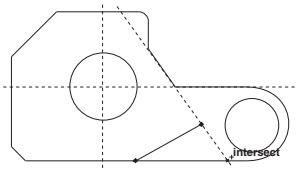
1. Add the cutout, beginning at the midpoint of the bottom edge and extending to the **-55°** construction line.
   - Select the **Connected Lines** tool.
   - Click at the ***midpoint*** of the bottom line of the part.



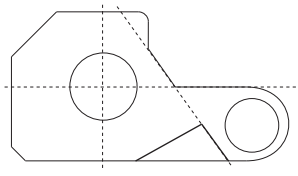
   - Move the pointer to the arc to display the ***center*** point of the arc.
   - Move the pointer along the **-55°** line until the ***intersection*** of the ***center*** point and the **-55°** line appears.



   - Click.
   - Move the pointer to the ***intersection*** of the **-55°** construction line and the bottom line of the drawing.



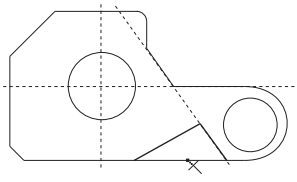
   - Double-click.



   The lines are drawn and remain selected.

2. Remove the unnecessary portion of the horizontal line.
   - Select the **Simple Trim** tool.

   - Position the **Trim** pointer dot over the line segment you want to discard.

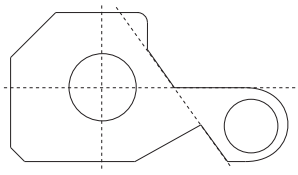


   - Click.

     The line segment is trimmed.

An object is automatically selected when its construction is finished. Both of these new lines are selected because you are using the **Connected Lines** tool, and both lines are the result of a single process. Once they are deselected, they are considered individual objects.

In order to use the **Simple Trim** tool, you must first select the lines that define the boundaries of the geometry you want to trim away, not the line you want to trim. In this case the lines are already selected.

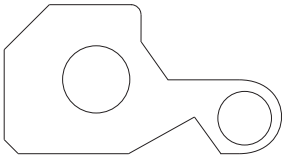


3. Delete the construction lines since you no longer need them.
   - Choose **Delete Constructions** from the **Layout** menu.

     The visible horizontal, vertical, and **-55°** construction lines created in this tutorial are removed. The *Drawing Assistant's* dynamic, on-the-fly construction lines continue to display when you move the pointer near geometry.
4. Change the **diameter** of the hole on the right to **3** cm.
   - Click the **Selection** tool.

   - Click the smaller circle.
   - Choose **Edit Objects** from the **Edit** menu.
   - Click the word **diameter** in the **Edit Objects** dialog box.
   - Type **3** and press **Enter**.

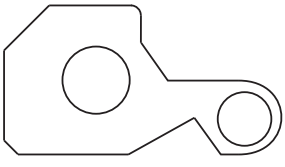
     If you are making only one change, you can press the **Enter** key; otherwise, you should not press **Enter**. First, make all the changes you want, then click the **Apply** button.

     The original circle is redrawn with a *3-cm* diameter.
   - Close the **Edit Objects** dialog box by double-clicking the **Close** box in the title bar.
5. Change the **thickness** and **color** of the lines.
   - Double-click the **Selection** tool.

     Everything in the drawing is selected.
   - Pull down the **Pen** menu and drag to display the **Weight** submenu.

     The arrow beside the **Weight** command on the menu indicates a submenu.
   - Choose **0,35**.

     All lines are changed to a medium pen weight.
   - Choose **Green** from the **Color** submenu of the **Pen** menu.
   - Click in the drawing area where there is no geometry, deselecting the lines so that you can see the new color.

     The selected lines don't change to green until you deselect them.

     The lines in the drawing are displayed in green.
6. Save your work.

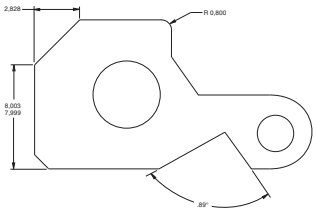
Accomplishments

- Trimming
- Deleting construction geometry
- Editing geometry with the **Edit Objects** command
- Selecting all objects
- Changing the pen weight
- Changing the pen color

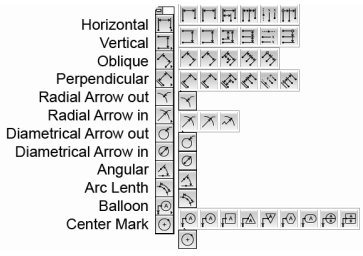
7

Dimensions

In this exercise you will add some dimensions to the drawing.



1. Dimension the horizontal length of the 45° line.
   - Choose **Show Palette** from the **Dimension** menu.



   The dimension palette appears

   - Select the **Horizontal dimension** tool.

   - Select the left end of the line to be dimensioned by clicking the lower endpoint of the 45° line.

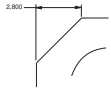
   The *hot spot* on the pointer moves to the right.

   - Click the upper ***endpoint*** of the 45° line.

   The dimension appears, but the text is in a location you may want to change.

   - Move the pointer to the dimension text to display the **Move** symbol.

   - ***Drag*** the text to the left horizontally until it is about **1** cm from the left leg of the dimension.

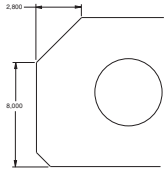


   The dimension text is repositioned.

2. Dimension the vertical line on the left.
   - Select the **Vertical dimension** tool.

   - Click the lowest point of the chamfered corner.

   - Click the upper ***endpoint*** of the vertical line.



Dimensions automatically go on the **Dimension** layer rather than on the work layer (except there exists no dimension layer or you have selected a different layer in the *Dimension Editor*). Only dimensions and construction lines go automatically on their respective layer. If you have removed the dimension layer, you will have to create a new one to complete this exercise. Since the layers are transparent, you see the geometry and dimensions on both layers at the same time.

Take a moment to examine the pointer that appears. The **dot** on one leg of the pointer marks the **hot spot** for the action you perform.

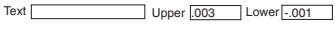
The location of the dot indicates which side of the object to select. The dot changes positions as you use the dimensioning tool. This type of pointer is a **smart pointer** because it gives you important information in a multistep process. Many of ***DraftBoard's*** tools use the smart pointer.

The order of the selection determines the placement of the dimension text. If you select in the order shown on the smart pointer, the text appears above horizontally dimensioned geometry or to the right of vertically dimensioned geometry. If you select in the opposite order, the text appears below or to the left of the selected geometry.

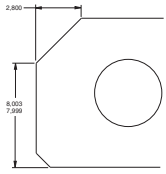
The dimension appears; however, you should add a tolerance.

- Select **Limits** from the **Preferences** of the **Dimension** menu.

A new group of status boxes appears at the bottom of the screen.



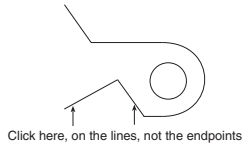
- Click the **Upper** status box.
- Type **.003** and press **Enter**.



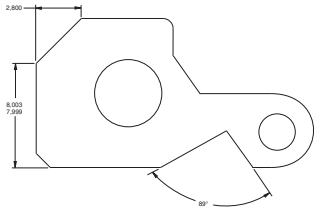
The dimension now reflects a **.003 upper tolerance** and **-.001 lower tolerance**.

3. Dimension the angle of the cutout.
   - Select the **Angular dimension** tool.

   - Click the arms of the cutout, near the lower ends of the lines.



   The angle is measured from the ***endpoint*** nearest the location you click.

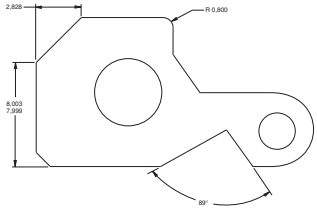


4. Add a radial dimension to the fillet.
   - Select the **Radial Arrow dimension** tool.

   - Choose **No Tolerances** from **Preferences** of the **Dimension** menu to return to dimensions without tolerance.

Radial dimensions are created with a single click of the mouse. The text appears on the side of the arc that you click.

   - Click outside (but near) the filleted corner. Make sure the ***on*** notation appears on the arc.



   The radial dimension appears on the side of the arc where you clicked.

   - Choose **Hide Palette** from the **Dimension** menu.
     The dimension tool palette disappears.

5. Remove the angular dimension.
   - Click the **Selection** tool
   - Click the text portion of the angular dimension
   - Press the **Back Space** key
     The angular dimension is deleted

6. Save this part with a different name (**part1a**), so that you can use it later for an advanced exercise.
   - Choose **Save As** from the **File** menu
     The **Save File** dialog box appears with **part1** listed in the File name box and the current file name part1.vlm in the entry box
   - Click after the **1** in **part1.vlm** in the entry box
   - Type **a** and press **Return**
     The part is saved with the new name

Accomplishments

- Adding dimensions
- Moving dimension text
- Adding tolerances
- Saving a version of a part with a different name

Crosshatching

8

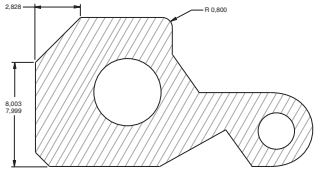
With the part fully drawn and dimensioned, you can now add crosshatching.

1. Select the boundaries that define the area to be crosshatched.
   - Double-click the **Selection** tool.
     All geometry is selected.

DraftBoard automatically excludes dimensions from the crosshatching process.

2. Crosshatch the part, indicating **Steel**.
   - Choose **Crosshatch** from the **Pen** menu.
   - Select **Steel** from the list of patterns.
     The display box shows the crosshatching exactly as it will appear in the part.
   - Click **Apply**.
     The part is crosshatched.
   - Close the dialog box.

You can use the **Hatch** command on the **Pen** menu to apply the current crosshatch pattern to the selected area.



   - Click anywhere in the drawing area to deselect the part.
3. Save the part once again as **part1**.
   - Choose **Save As** from the **File** menu.
     The file name listed is **part1a.vlm**.
   - Click after the **a** in the entry box.
   - Press the **Back Space** key once to remove the **a** and press **Enter**.
     Since you can't save two documents with the same name in the same directory, you are asked if you want to overwrite the existing file named **part1.vlm**.
   - Click **OK**.
     The original version of **part1** is replaced with this crosshatched version.

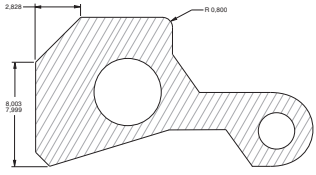
Accomplishments

- Crosshatching a part
- Saving and replacing an existing version
- Renaming a file

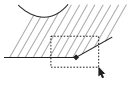
9

Stretching

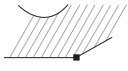
In this exercise, you will change the basic outline of the part and see how the crosshatching automatically redraws to accommodate the change.



1. ***Drag*** a selection marquee around the point where the left point of the cutout joins the horizontal line.
   - Click the **Selection** tool.
   - If any part of the drawing is selected, click anywhere in the drawing area to deselect the part.
   - Position the pointer above and to the left of the point.
   - ***Drag*** to a location below and to the right of the point.

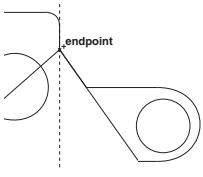
     The point is selected, as shown below. If you don't see the square selection point, choose **Selectable Points** from the **Edit** menu and select again.

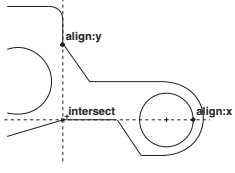
2. Stretch the part so that the left side of the cutout is horizontal.
   - Move the pointer to the selected point until the pointer displays the ***Move*** symbol.

   - ***Drag*** the point upward to the top of the **-55°** line, so that the vertical construction line appears. ***Do not release the mouse button!***.

     Dragging the pointer over this ***endpoint*** activates it so that the Drawing Assistant's construction line passes through it.

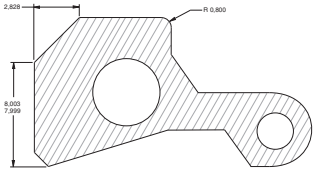
     

   - With the mouse button still pressed, ***drag*** downward to the ***intersection*** of the Drawing Assistant's construction lines as shown below.

   - Release the mouse button.

     The part is redrawn, and the crosshatching is updated.

3. **Save** your work.

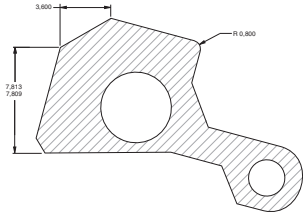
Accomplishments

- Selecting a point
- Using a selection marquee.
- Activating a point
- Stretching a part

Rotating

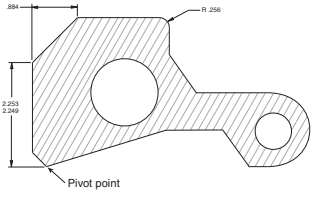
10

Now that the part is complete, you can rotate it so that the left line of the newly created cutout is horizontal.



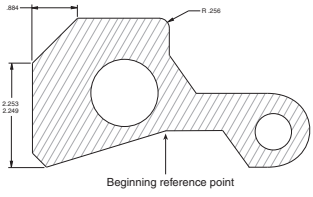
1. Specify the rotation.
   - If the part is not selected, double-click the **Selection** tool.
   - Select the **Rotate** tool from the **Transformation** subpalette.

   - Specify the *pivot point* (the center of rotation) by clicking the lower ***endpoint*** of the left side of the cutout.

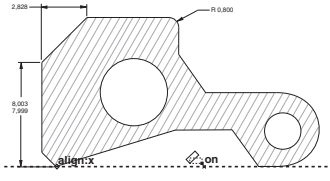
     

   - Specify the beginning **Reference point** by clicking at the control point you moved in the last exercise.

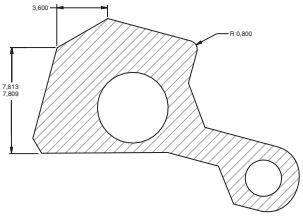
     The beginning **Reference Point** is the point that is to move)

   - Specify the **ending reference point** by clicking on the horizontal construction line across the bottom of the part.

   The part rotates.

Note that the dimensions changed to reflect the new orientation; this is an example of associativity. ***DraftBoard*** maintains an extensive database describing all geometry so that dimensions and locations can be updated quickly.

15

2. Reduce the visual display of the part.
   - Select the **Zoom Out** tool to display the entire part on the screen.

     The **Scale** box in the status line displays the current scale.
   - Click a location near the larger hole.

     The magnification of the part is reduced and the location you clicked is in the center of the screen.

Zooming does not change the actual measurements of the part.

3. Save your work.

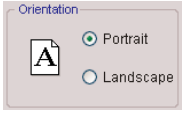
Accomplishments

- Rotating a part
- Observing associative dimensions
- Zooming out to a specific location

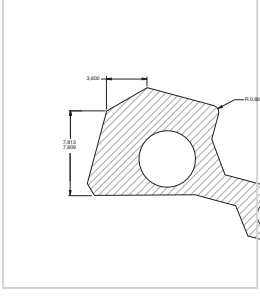
11 Printing

For your drawing to be useful, you need to transfer it to paper. If your drawing extends past the boundaries of the paper you are using, you can scale it before printing.

1. Specify the page orientation and paper size.
   - Choose **Print Setup** from the **File** menu.
   - If necessary, specify **Portrait** orientation.

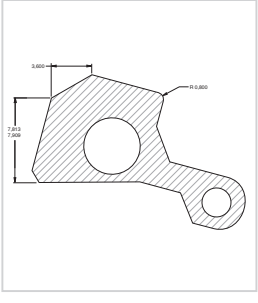
   - If you have a plotter, specify the appropriate paper size.
   - Click **OK**.
2. Specify the exact area to be printed.
   - Select **Drawing Size** from the **Layout** menu.
   - Click **Always Display** to display the gray box representing the maximum plotable area of the page.

When you select the option **Keep Text Size** and **Keep Dimension Text Size**, text and dimensions are not scaled but keep their original text size.

The page outline appears in the drawing area at all times. When this option is off, the page outline appears only when the **Drawing Size** dialog box is open or you can select **Preview Layout** from the **File** menu.

   - Click **Fit**.

Your drawing may not look like the right illustration because you may have specified a different printer, plotter, or paper size.

   The drawing border is scaled and redrawn so that the part fits on the paper size with the orientation you specified.

- Click **OK** to leave the dialog box.

3. Print the drawing.
   - Choose **Print** from the **File** menu.
     The drawing is sent to the printer or plotter.
4. Close the document.
   - Choose **Close** from the **File** menu.
     You are asked if you want to save.
   - Click **OK**.
     The document is closed. When you open another ***DraftBoard*** *document*, a new ***DraftBoard*** *window* appears.

Accomplishments

- Displaying the paper size in the drawing area
- Scaling the drawing
- Printing the drawing

CHAPTER

16

Advanced Features

Trimming and Relimiting

Text

Tangent and Perpendicular Lines

Rotational Copy

Constructing a Side View

Advanced Crosshatching

Mirror Images and Bolt Circles

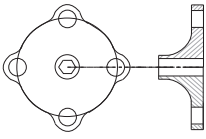
NURB Splines

Creating a Detail View

Drawing to Scale

Advanced Features

In this section, you use some of the more advanced ***DraftBoard*** features. While many features are demonstrated, you should look through the *User Guide* chapters for other features which might be useful to you. In some cases, the exercises do not create a useful part but only show you how to use a feature, such as how to trim two lines to make a corner or how to construct a line tangent to a circle. In other cases you create real parts, such as the front and side view of a flange.



The features covered in this section include:

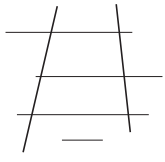
- Corner trim
- Trim
- Relimit
- Text
- Tangent lines
- Perpendicular lines
- Origin (0,0)
- Circle fillets
- Polar duplicate
- Polygon
- Parallel lines
- Mirror transformation
- Bolt circle
- Splines

1 Trimming and Relimiting

In this exercise, you will investigate some advanced construction techniques. ***DraftBoard*** allows you to create corners and trim or re-limit lines. When you trim a line it is shortened to its intersection with the selected boundary. Relimiting allows you to extend or shorten a line to the limiting boundary.

1. Open a new ***DraftBoard*** document.
   - Choose **New** from the **File** menu.
2. Explore the **Trim** tool.
   - Create an approximation of the lines below.

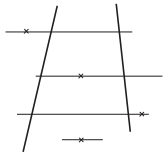
The exact figure is not important. You are only learning about the functions of the **Trim** and **Relimit** tools.



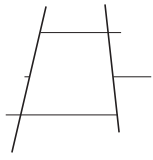
- Hold down the **Shift** key and select the two lines that appear bold in the illustration above.
- Select the **Trim** tool.

If you trim on entire entity, a prompt will come up asking if you want to delete the entire item. ***DraftBoard*** is making sure you want to remove that piece of geometry before it performs trim.

- Click the locations indicated below.



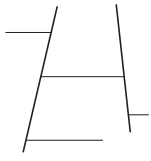
The lines are trimmed.



For **Trim**—select what you want to throw away.

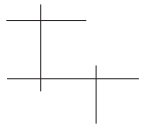
For **Relimit**—select what you want to keep.

3. Explore the **Relimit** tool.
   - Select **Undo** from the **Edit** menu four times so that the lines are restored.
   - Click the same locations with the **Relimit** tool.



The lines are extended to the boundaries.

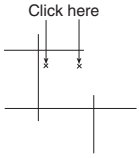
4. Explore the **Corner** tool.
   - Create an approximation of the lines below.



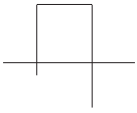
- Double-click the **Selection** tool to select all lines.
- Select the **Corner Trim** tool from the **Trim** subpalette.

You can either click each line individually or use **Shift**-Click, clicking inside the corners.

- Hold down the **Shift** key and click inside the *top corners* as shown below.



The lines are trimmed to create corners.



If you make mistakes, you can use **Undo** and **Redo** to go back to a previous version.

- Select the **Trim** tool.

- Click the parts of the lines to be trimmed away (those that extend past the corners) to create the figure below.

5. Delete all geometry.
   - Choose **Select All** from the **Edit** menu.
   - Press the **Back Space** or **Delete** key.

Accomplishments

- Using the Corner trim tool
- Observing the difference between trim and relimit
- Deleting geometry

Text

2

In this exercise, you will explore using text.

1. Create a text block.
   - Choose the **Text** tool.

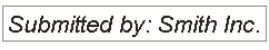
   - ***Drag*** a box in the center of the screen—the size isn't important.

   - Click the **Width** status box.
   - Type **3** and press **Enter**.

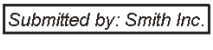
Once you leave the **Text** tool, you can change the size of the text-entry box by ***dragging*** a selection marquee around the *control points* on the right side of the box and ***dragging*** the points. Control *Points* are the points defined when you first created the text block and represent the endpoints of the text block.

Instead of **10-point Times** you can use any font listed in the **Font** submenu of the **Text** menu.

2. Change the text characteristics to **10**-point **Times** *italics*.
   - Choose **Times** from the **Font** selections on the **Text** menu.
   - Choose **10** from the **Size** selections on the **Text** menu.
3. Type this text: **Submitted by: Smith Inc.**
   - Press **Enter**.



4. Draw a box around the text.
   - Select the **Rectangle** tool.
     The **Text** entry box disappears.
   - ***Drag*** a box around the text so that the text is centered within the box.



The text box appears only when the text is selected. The box you are drawing now will always be visible.

5. Group the box and the text so that they can be treated as a single unit.

When you want several entities to act as one, **group** them. In that way you won't accidentally select one, when you meant to select the components of the group.

- Double-click the **Selection** tool.
- Choose **Group** from the **Arrange** menu.
- ***Drag*** the box around to see that the text and the box act as a single entity.

6. Type a list of notes.
   - Create another text box, **5** cm wide.
   - Change the text characteristics to accommodate a plotter:

     Font: **Plotter**

     Size: **3,5 mm**

     Style: **Normal**
   - Type the following without pressing **Enter**:

     1. The materials list is included on a separate sheet.

     1. The materials list is included on a separate sheet.

Use the **Delete** key to make corrections as you type. Also, notice *word wrap*—the text automatically wraps to the next line when a word extends past the right margin.

   - Press **Enter** to begin a new line.
   - Type the following:

     2. Tolerances are specified as needed.

     1. The materials list is included on a separate sheet.
     2. Tolerances are specified as needed.

7. Change the indentation so that the text aligns under itself rather than under the number.
   - Use the **Selection** tool to select the note text.
   - Choose **Indentation** from the **Text** menu.
   - Click the entry box **Left Indent** to highlight it.
   - Type **0,9**.
   - Click **OK**.

     1. The materials list is included on a separate sheet.
     2. Tolerances are specified as needed.

     The text shows a hanging indent.
8. Make changes to the text.
   - With the text still selected, click the **Text** tool.
   - Double-click the word **materials**.

     The word *materials* and the *space* after it are highlighted.
   - Type **specifications** and press the **Spacebar** once.

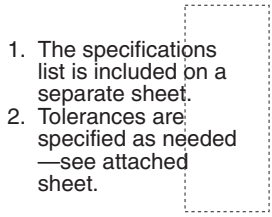
     1. The specifications list is included on a separate sheet.
     2. Tolerances are specified as needed.
   - Click after the period that ends the second sentence.
   - Press the **Back Space** or **Delete** key.

     The period is deleted.
   - Press the **Spacebar** once and type the following:

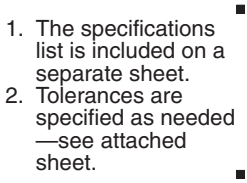
     —see attached sheet.

     1. The specifications list is included on a separate sheet.
     2. Tolerances are specified as needed —see attached sheet.

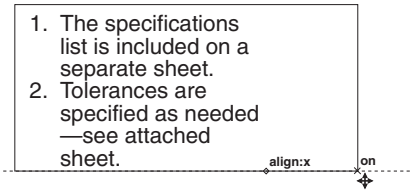
9. Change the size of the text box.
   - Click the **Selection** tool.
   - ***Drag*** a selection marquee around the right side of the text box.



   The upper and lower corner points of the box are selected.



   - ***Drag*** the *control points* about **2 cm** to the right.



   The area is resized and the text is redrawn.

1. The specifications list is included on a separate sheet.
2. Tolerances are specified as needed—see attached sheet.

The *Text Margin* (the distance between the text and the text box) can be edited with the **Text Margin** command from the **Preferences** menu in the **Layout** menu. The text margin can be edited as well using the **Edit Objects** command from the **Edit** menu.

10. Explore the text processing functions until you feel comfortable with the tools, then delete all the text you created in this exercise.
    - Choose **Select All** from the **Edit** menu.
    - Press the **Back Space** key to delete all selections.

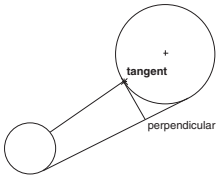
Accomplishments

- Entering text
- Changing text characteristics
- Grouping entities
- Specifying a hanging indent
- Making changes to text
- Changing the size of the text area

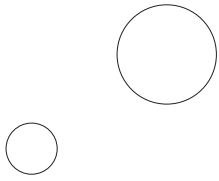
Tangent and Perpendicular Lines

3

In this exercise, you will explore drawing tangent and perpendicular lines.

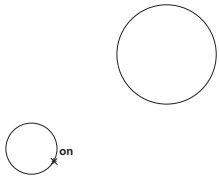


1. Draw two circles like those shown below.



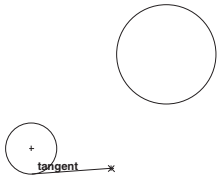
2. Construct a line tangent to the lower edges of both circles.
   - Click the **Single Line** tool.
   - Move the pointer to the circle on the left until the ***on*** notation appears.

The *Drawing Assistant* must display ***on*** and not other circle notations such as ***quadrant***.



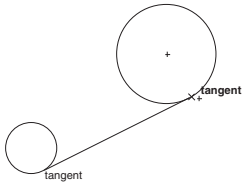
   - Press and hold the mouse button and move the pointer away from the circle at approximately a 45° angle until the ***tangent*** notation appears as shown below.

Notice that the line you ***drag*** stays tangent to the circle wherever you ***drag*** it.

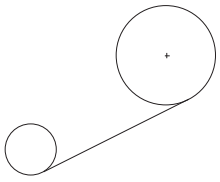


   - ***Drag*** the pointer to the lower edge of the large circle until the ***tangent*** notation appears.

As you have seen, tangent lines can be pulled off circles and other lines, as well as ellipses, splines, and arcs.



   - Release the mouse button.
     The tangent line is drawn.

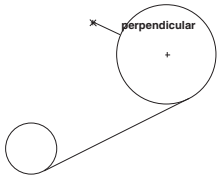


3. Construct a line perpendicular to both circles.

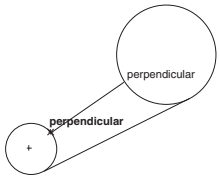
The *Drawing Assistant* must display ***on*** and not other circle notations such as ***quadrant***.

   - Press the mouse button when ***on*** appears on the large circle.
   - ***Drag*** directly away from the circle at approximately a 90° angle until the ***perpendicular*** notation appears.

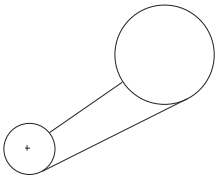
Notice that the line you ***drag*** stays tangent to the circle wherever you ***drag*** it.



   - ***Drag*** to the left circle until the ***perpendicular*** notation appears.

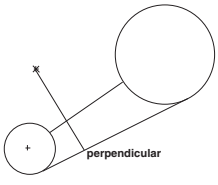


- Release the mouse button.

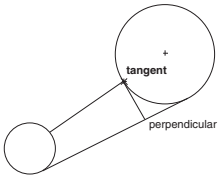


The line is drawn perpendicular to both circles.

4. Construct a line perpendicular to the lower straight line and tangent to the larger circle.
   - Press the mouse button when ***on*** appears on the lower straight line.
   - ***Drag*** at a **90°** angle from the line to display the ***perpendicular*** notation.



   - ***Drag*** the new line along the baseline to the large circle until ***tangent*** appears and release the mouse button.



The line is drawn tangent to the large circle and perpendicular to the lower straight line.

5. Save or discard the file, as you wish.

As you have seen, tangent and perpendicular lines can be pulled off circles and other lines, as well as ellipses, splines, and arcs.

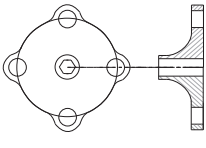
Accomplishments

- Constructing tangent lines
- Constructing perpendicular lines

Rotational Copy

4

In the next four exercises, you will construct a flange with a side view.

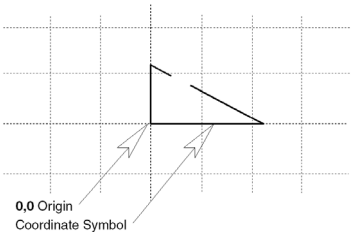


For this exercise, draw a flange **9 cm** in diameter containing four **1,5 cm** lugs each with **0,75 cm** holes. Center a **1 cm** hexagonal hole in the **1,5 cm** hub of the flange.

1. Open a new document.
2. Choose a different pen style.
   - Choose **Visible** from the **Pen** menu.

     The pen style is now solid, black lines **0,5** mm wide.
3. Draw a **9 cm** circle centered at **0,0**.
   - Choose **Show Grid** from the **Layout** menu, so that you can see the origin.

     The grid appears on the drawing area.



- Choose the **Center-Point Circle** tool.
- Type **9** in the status line and press the **Tab** key.
- Type **0** and press **Tab**.
- Type **0** and press **Enter**.

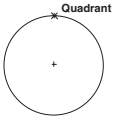
  The *9 cm* circle is drawn, centered at 0,0.
- Choose **Hide Grid** from the **Layout** menu.

  The grid and **Coordinate symbol** disappear.

4. Draw a *0,75 cm* lug hole centered at the top of the *9 cm* circle.

   The **Center-Point Circle** tool is still the current tool.

   - Click on the 12 o'clock ***quadrant*** notation.

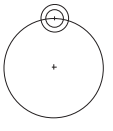


   - Enter **0,75** in the status line and press Enter.

     A **0,75-cm** circle is drawn, centered on the top of the **9-cm** circle.

5. Draw a *1,5-cm* circle with the same center as the *0,75-cm* circle.

   - Enter **1.5** and press **Enter**.



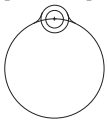
     Another circle is drawn, centered at the same place and with a diameter of *1,5 cm*.

6. Create fillets with the default 0,5-cm radius where the lug joins the flange.

   - Click the **Fillet** tool.

   - Hold down the **Shift** key and click between the two curves, as shown by the **x** in the illustration below.

   - Repeat the process on the other side of the lug.



     The fillets are complete.

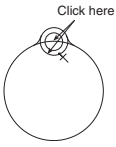
7. Trim the lug.

   - Click the **Selection** tool.

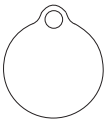
     The last fillet you created is still selected.
   - Hold down the **Shift** key and click the other (unselected) fillet.

     Both fillets are selected to be used as the boundaries for trimming.
   - Click the **Simple Trim** tool.

   - Click the bottom of the **1,5-cm** circle and the top of the **9-cm** circle.

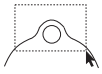


The circles are trimmed between the fillets.



8. Construct a total of four lugs for the flange.
   - Click the **Selection** tool.
   - ***Drag*** a marquee around all entities that make up the lug.

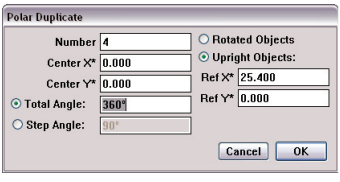
     Only the objects inside the marquee are selected.

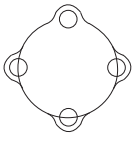
     The lug is selected.
   - Choose **Polar Duplicate** from the **Edit** menu.

     The dialog box should be set to rotate **4** objects, with the **x,y** coordinates set at **0,0**.

     The **asterisks** in the dialog box indicate that you can enter coordinates into the **Center X** and **Center Y** boxes or you can specify a location with the mouse. To use the mouse, first click the **Center X** box. Then click the location for the center on the drawing area. For this exercise, you want the default location of **0,0**, so you don't need any entries in this box.

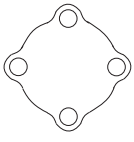
   - Click **Ok**.

     The lug is copied, but the 3-inch circle must be trimmed.

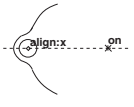
9. Trim the circle inside the copied lugs.
   - Use the **Selection** tool and the **Shift** key to select the fillets for the three copied lugs.

     If you click something you don't want to select, continue holding down the **Shift** key and click again to deselect it.
   - Choose the **Trim** tool and trim the circle between the fillets.

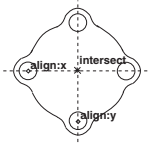
     

     The lugs are complete.

10. Construct a **2-cm** circle in the center of the flange.
    - Select the **Center-Point Circle** tool.
    - Move the pointer to display the horizontal construction line through the left bolt hole.

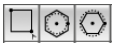
      

    - Display the vertical construction line for the lower bolt hole and click at the intersection of these two construction lines.

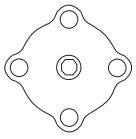
      

    - Enter **1,5** and press **Enter**.

11. Add the **1-cm** (outside measurement) hexagonal hole within the **1,5-cm** hub.
    - Select the **Inscribed Polygon** tool.

      Confirm that the status line shows the **x**,**y** coordinates as **0,0**. If this is not the case, enter 0 into each of these boxes.
    - Type **1** in the Diameter box (do not press **Enter**).
    - Press the **Tab** key to select the **Sides** box.
    - Type **6** and press **Enter**.

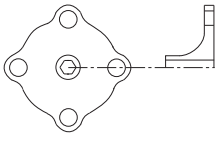
      The hexagonal hole is drawn and the flange is complete.
12. Save the part, naming it **flange**.
    - Choose **Save** from the **File** menu.
    - Type **flange** and press **Enter**.

Accomplishments

- Displaying the grid
- Placing geometry at the origin
- Filleting circles
- Making multiple selections
- Trimming unnecessary geometry
- Rotating and copying the lug
- Constructing an inscribed hexagon

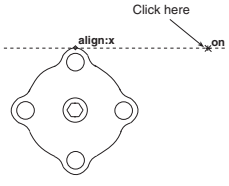
5 Constructing a Side View

Create a side view of the flange that is 3 cm thick at the hub and *0,5 cm* thick at the lugs.

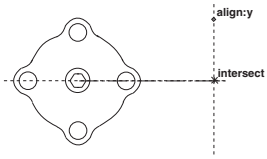


1. Zoom the part.
    - Click the **Zoom Out** tool.

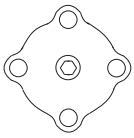
    - Enter **0,5** in the status line.
2. Construct a vertical line for the side view that is the same length as the distance from the top of the flange to the center.
    - Click the **Single Line** tool.
    - Move the pointer to the top of the flange to display the construction line and move to the right and click ***on*** the construction line as shown.

    - Move the pointer to the center of the right bolt hole to display the horizontal construction line, then move to the right and click at the ***intersection*** of the horizontal and vertical construction lines.



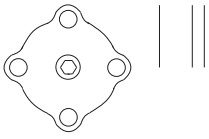
The first vertical line for the side view is drawn.



3. Construct the remaining vertical lines of the side view.
   - Click the **Parallel Line** tool.

   - ***Drag*** a line to the right of the vertical line in the side view.
   - Enter **2,5**.

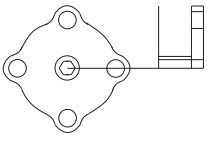
     The parallel line is drawn **2,5** cm to the right of the original line.
   - Click the original vertical line.
   - Enter **3**.

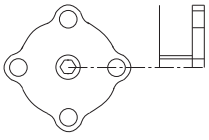
     The vertical lines are complete.
4. Construct the horizontal lines for the side view.
   - Select the **Single Line** tool.
   - Draw a horizontal line across the top of the right arm of the side view.

   - Continue drawing the horizontal lines aligned with these locations as shown in the illustration below:

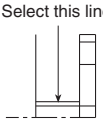
     Top of bolt hole
     Bottom of bolt hole
     Top of hub
     Top of hexagon
     Center of flange

     The lines are drawn and the centerline is selected. If it is not, use the Selection tool to select the line from the center of the flange to the side view.
   - Choose **Center** from the **Pen** menu.

     
   - Select the **Selection** tool.
   - Click outside the flange to deselect everything.
   - Choose **Visible** from the **Pen** menu.

5. Trim the excess lines.
   - Save the drawing in case things don't go as expected.
   - **Zoom In** on the side view only.
   - Select the trim boundary.

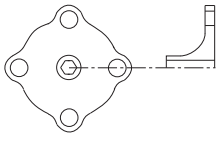
     

   - Trim the vertical line on the left that extends above the selected horizontal line.

     The vertical line is trimmed.
   - Trim the vertical line that descends from the inside corner to the centerline.

     The trimming is complete.
   - When the drawing looks right, save it again.
6. Create the hub fillet with a radius of 1,5 cm.
   - Select the Fillet tool.
   - Enter **1,5** in the **Radius** box.
   - Hold the **Shift** key down and click inside of the corner of the side view.

     The fillet is redrawn to a **1,5-cm** radius.

     
   - **Zoom out** to see the entire flange.

You can use the **Shift** and **Ctrl** keyfor **Stroke zoom**, *dragging* from the upper left to the lower right across the side view. You can also select the **Zoom In** tool and ***drag*** a marquee around the side view. If zooming takes you places you didn't want to go, choose **Zoom All** from the **Arrange** menu and all geometry appears and fills the window.

If you have had trouble and the drawing is not right, close this document without saving it. To close without saving, double-click the *Close box* icon, then click **No** when asked if you want to save. Next, choose **Open** from the **File** menu and open the flange document again and repeat *Exercise 5*.

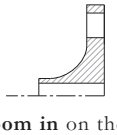
Accomplishments

- Constructing a side view
- Creating parallel lines
- Creating a centerline
- Trimming corners

6

Advanced Crosshatching

In this exercise, you will crosshatch the side view. Since the side view isn't a single closed figure, you will have to segment some of the lines for the crosshatching to work properly.

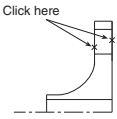


You can also use the **Tracer** Tool to select these boundaries unsegmented. You might practice both methods of selecting lines.

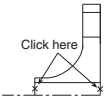
1. **Zoom in** on the side view.
2. Select all geometry for the side view by ***dragging*** a selection marquee around the side view.
3. Use the **Segment** Tool to break the vertical lines at the ***intersection*** of the horizontal lines. This is necessary to define a closed boundary to crosshatch.
   - Choose the **Segment** tool from the **Trim** palette.

To understand this process, consider how you created the side view. The right vertical line is a single line which is met by the horizontal lines forming the bolt hole and center hole. *DraftBoard* cannot determine the boundary because the horizontal lines touch the vertical lines within the closed figure. In this exercise, you'll segment the boundary lines so they can be selected to form two closed shapes.

- Click the vertical lines to be segmented.

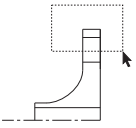


- Click the lines representing the center hole.

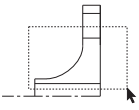


Although no visual change has occurred, the vertical lines have been segmented-broken into shorter, connected lines.

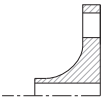
4. Crosshatch the solid sections of the side view.
   - Click the **Selection** tool.
   - Click anywhere in the drawing area, to deselect the side view.
   - ***Drag*** a selection marquee around the top part of the side view.



   - Hold down the **Shift** key and ***drag*** a selection marquee around the lower solid section of the side view.



   Both sections are selected.
   - Choose **Crosshatch** from the **Pen** menu.
     The **Iron pattern** is already selected and displayed in the pattern box.
   - Double-click the **Spacing** box.
   - Enter **0,2** and click **Apply**.
   - Close the dialog box.



   The side view is crosshatched.
5. Save the part.

Accomplishments

- Segmenting lines
- Crosshatching complex figures

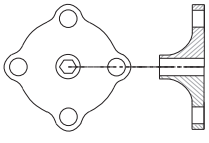
Mirror Images and Bolt Circles

7

In this exercise, you will create the bottom half of the side view and add a bolt circle to the front view of the flange.

1. Create the bottom half of the side view.
   - ***Drag*** a selection marquee around the entire side view.
   - Select the **Mirror Transformation** tool.

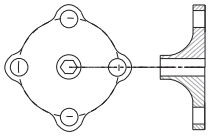
   You are asked to specify a reference line.
   - Hold down the **Ctrl** key and click ***on*** the centerline at two places in the side view.



Ctrl-click causes the **Transformation** tool to make a copy and transform the geometry.

The side view is complete.

2. Add a bolt circle to the front view.
   - Select the **Selection** tool and click anywhere to deselect the side view.
   - Choose **Center** from the **Pen** menu.
   - Select the **Center-Point Circle** tool.
   - ***Drag*** from the center of the flange to the center of a bolt hole.



   - Use the **Selection** tool and click in the drawing area to deselect the circle.
   - Choose **Visible** from the **Pen** menu.
3. Save the part and close the document.
   - Close the current ***DraftBoard*** window.

     You are asked if you want to save the document.
   - Click **Yes**.

     The document is saved and its window closed. You may see other windows, if other ***DraftBoard*** documents are open.

Accomplishments

- Creating a mirror image
- Creating a bolt circle

8

NURB Splines

In this exercise, you will create and edit splines and observe the difference between splines constructed through specific points and those constructed from vectors.

1. Open a new document.
2. Select the **Through-Points Spline** tool.

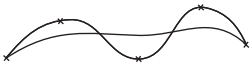
3. Click points such as those shown below, double-clicking the last point.

   A ***NURB*** *spline* is drawn through the *control points* you specify.

4. Select the **Vector Spline** tool.

5. Click the same points (the ***vertex*** notation appears at the points you clicked for the first spline).

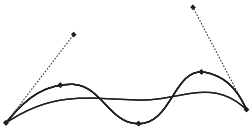
   A different curve is constructed using the vectors for calculation.



6. Edit the *control point* in the middle of the original spline.
   - Select the original spline.

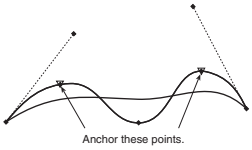
- Choose **Show Points** from the **Layout** menu.

  The *control points* and the *endpoint slope controls* are displayed.



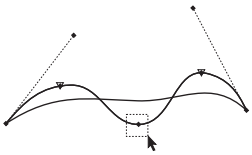
- Select the **Lock Spline Control Point** tool.

- Click the *control points* on either side of the point you want to change.



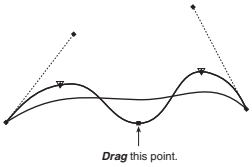
The points are locked.

- Deselect the spline.
- Select the *middle control point*.



If you can't select this point, choose **Selectable Points** from the **Edit** menu and reselect the *control point*.

- ***Drag*** the middle *control point* to new locations and observe how the spline is redrawn.



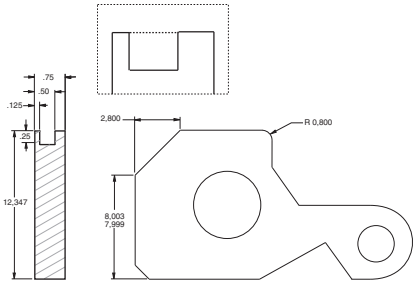
Accomplishments

- Creating a through-points spline
- Creating a vector spline
- Editing splines

Creating a Detail View

9

In this exercise, you will create a detail view. A detail view is an enlarged or reduced view of part of the drawing. Detail views are dynamically linked to the part from which they are derived, and modifications made to the part are also made to the detail view. This functionality is called associativity.



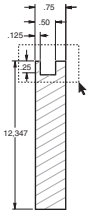
1. Create a detail view of the notch from *Exercise* ***8*** in the last section of the tutorial, using a **2-to-1** ratio.

   - Open the file named **part1a**.

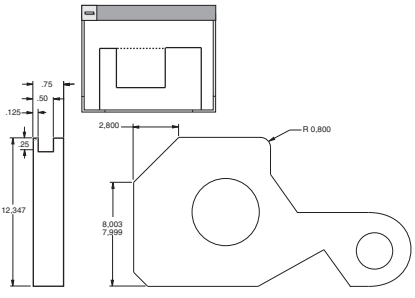
- Click the **Detail View** tool on the **View Control** subpalette.

The **Scale** box is highlighted in the status line.

- Enter **2**.
- ***Drag*** a frame around the top of the side view, including the notch.



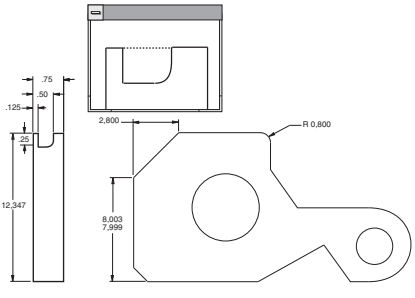
- ***Drag*** the frame to a new location.



Note that the detail view is not dimensioned. *Crosshatching*, *dimensions*, and *text* are view dependent—they appear only in the view in which they were created according to standard drafting practice.

The enlarged geometry appears in the detail view frame.

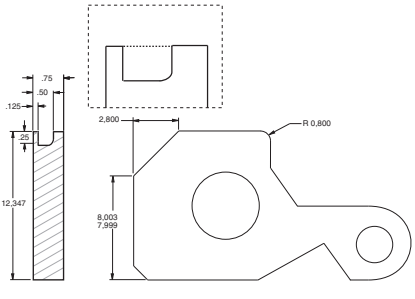
2. Fillet the right corner of the notch in the detail view and observe that the geometry in the frame is, indeed, a view of the same part.
   - Click the **2-Entity Fillet** tool.
   - Enter **0,25**.
   - Select the two lines that form the right corner of the notch in the detail view.



If you want to know more about working with views, see the *Views* chapter in the *User Guide*.

Both the part and the detail view are redrawn with the fillet.

3. Crosshatch the side view.
   - Deselect the view by clicking outside the view with the **Selection** tool.
   - **Crosshatch** the side view with the **steel** crosshatch pattern.



The side view is crosshatched but the detail view is not.

4. Save or discard the file, as you wish.

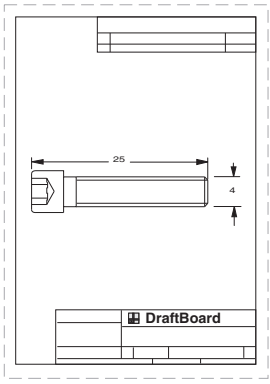
Accomplishments

- Creating a detail view
- Observing associativity

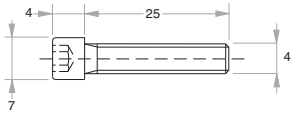
10

Drawing to Scale

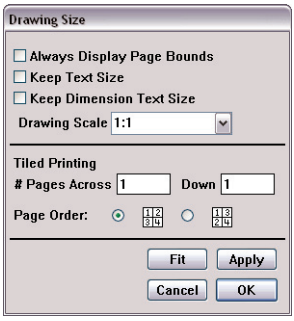
In this exercise you will use the Symbol feature to display a full-scale part. Then you will dimension and scale the part for placement on a standard drawing format. This exercise uses a simple part to demonstrate the procedure you should use to construct more complex full-scale parts. Drawing at full scale has several advantages, the most important of which is that you can start your work without having to specify paper formats or other parameters which only would distract from the design task.



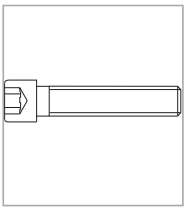
1. Draw a side view of a Phillips-head screw which is *25 mm* long and has a thread diameter of *4 mm*.
   - Open a new drawing using the **New** command in the **File** menu.
   - Draw the side view of the screw using the drawing tools in the tool palette.



2. Set the specifications for the size paper you will be using.
   - Choose **Print Setup** from the **File** menu.
   - Select the paper size **DIN A4** for a laser printer or **a** size **A1** for a plotter.
3. Set the visual scale of the geometry.
   - Choose **Drawing Size** from the **Layout** menu.
     The Drawing Size dialog box is displayed.



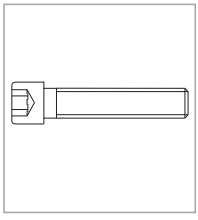
- Click **Fit**.



The screw is scaled to fit the paper size.

The scale is approximately **7:1** but the geometry is visually too large for the page, so you should reduce the scale to **6:1**.

- Enter a scale of **6:1** and click **OK**.



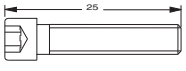
The size of the geometry is now more appropriate for the size of the page boundary.

- Click **OK**.

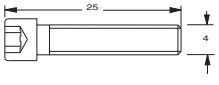
4. Dimension the screw.

Zooming a part does not change its measurements, it magnifies or reduces it to display the entire part on the screen. In this way you can draw a part full scale, whether it is **100 meters** long or **1 millimeter** wide.

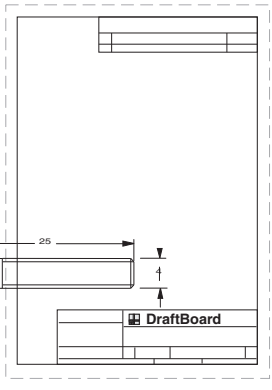
- Choose **Zoom All** from the **Arrange** menu to display the screw at a larger size.
- Choose the **Horizontal dimension** tool.
- Click the opposite ends of the screw.
  The dimension displays



- Dimension the diameter of the thread.

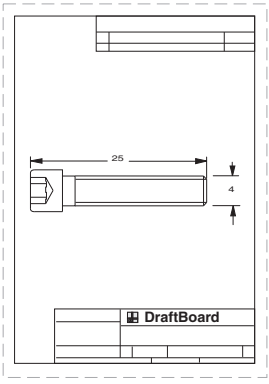


5. Place the scaled drawing into a standard A-size drawing format.
   - Choose **Zoom Out** from the **Arrange** menu.
   - Choose **Import** from the **File** menu.
   - Open the **Layouts** directory.
   - Click the **aFormatA** file in **Layouts** directory.
   - Click **Unscale** and **OK** to import the unscaled drawing format.



The format is imported to the sheet and is selected

- ***Drag*** the format until the screw is properly positioned on it.



6. Plot the drawing.
   - If your plotter or printer is set up and if you want a copy of this drawing, choose **Print** from the **File** menu.

     The drawing is plotted.
7. Save or discard the file, as you wish.

Accomplishments

- Using a symbol
- Scaling a drawing
- Importing a drawing format.

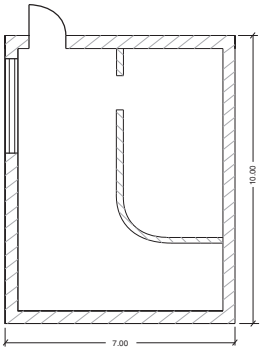
CHAPTER

17

Architectural Drawings

Architectural Drawings

In this section, we will create a simple two-dimensional architectural drawing. Using the ***Smart Wall*** *tool* and the ***Smart*** *Symbols* you can sketch a architectural drawings as fast as using paper and pencil

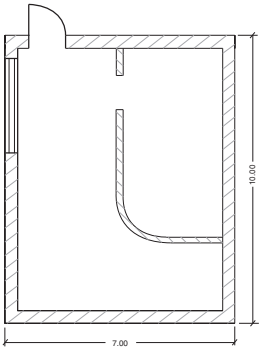


The features covered in this section include:

- Drawing Smart Walls
- Placing Smart Symbols
- Scaling Drawings
- Printing Drawings

1 Creating an Architectural Drawing

In this exercise, you will use the **Smart Wall** tool to create a simple architectural drawing, a floor plan with outside measurements of **7** meters by **10** meters.



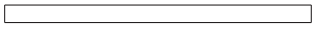
It is critical that you hold down the **Shift** key and ***drag*** to construct this wall so that the wall thickness is not added to the length of the first wall.

1. Open a new DraftBoard document.
2. Set the units to **Meters**.
   - Choose **Units** from **Preferences** in the **Layout** menu.
   - Click **Meters** and click **OK**.

You can use different units of measure in the status line entries. If the selected units is not **Meters**, be sure that you specify the unit such as 2 **m** for two **meters** or 2 **cm** for two **centimeters**.

3. Create two new layers: one for the interior walls and one for the exterior walls.
   - Select **Layers** from the **Layout** menu.
     The **Layers** dialog box appears.
   - Click **New** to create another layer.
   - Rename the layer by typing **Interior walls** in the **Rename** entry box and click **Rename**.
   - Click **New** to create another layer.
   - Rename the layer by typing **Exterior walls** in the **Rename** entry box and click **Rename**.
   - Select **Exterior walls** and click **Current**, to make **Exterior walls** the current layer.
4. Draw the exterior walls
   - Select the **Smart Wall** tool from the **Line** subpalette.

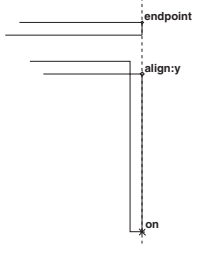
   - Enter **24 cm** in the **T** status box to indicate walls that are **24** cm thick.
   - ***Drag*** a horizontal line.
   - Enter **7** in the **L** status box and press **Return**.
     The line extends off the screen.
   - Choose **Zoom All** from the **Arrange** menu.

     The entire line is visible.

Click twice on the down arrow to scroll the horizontal wall upward on the screen.

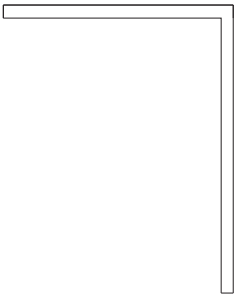
   - Press and hold the **Shift** key and ***drag*** down vertically from the ***endpoint*** of the line at the upper-right corner.



It is critical that you hold down the **Shift** key and ***drag*** to construct the wall so that the wall thickness is not added to the length of the first wall.

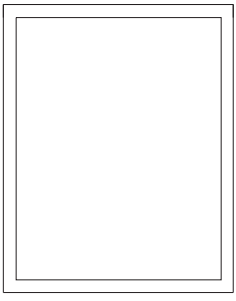
   - Enter **10** and press **Enter**.
     A corner is made between the first and second line, and the second line extends off the screen.

- Choose **Zoom All** from the **Arrange** menu.



Selecting **Show Points** in the **Layout** menu helps you to align the defining points of a wall precisely to the defining points of the previous wall segment.

- Complete the rectangle, using the **Shift** key while ***dragging*** the third wall.



In the **Edit Objects** dialog box you can change the defining axes of selected walls.

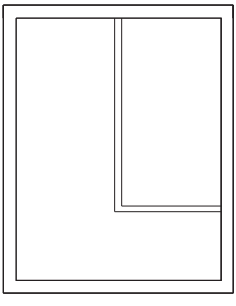
5. Change the current work layer with the **pop-up** layer box at the lower left of the drawing area.
   - Click anywhere in the drawing area, to deselect all walls.
   - Press the mouse button on the box and the menu displays.
   - ***Drag*** to **Interior walls** and release the mouse button.

     Interior walls is now the current work layer.
6. Create a drawing like the one below with two interior walls (10 cm thick) which don't merge with the exterior walls.
   - Move the pointer along the inner edge of the upper horizontal wall until the notation ***midpoint*** displays.
   - ***Drag*** a wall segment about 6 m long.
   - Enter **10 cm** for the thickness of the wall in the status line and press the **Tab** key.
   - Enter **6** in the **L** status box and press **Return**.

     The horizontal wall is not merged with the vertical wall segment, since they are not on the same layer.

Only walls on the same layer will be merged.

   - ***Drag*** a horizontal wall to connect the vertical wall segment with the right vertical exterior wall.



7. Construct a **1 m** hole through the vertical interior wall, **1 m** distant from the upper horizontal wall.
   - Move the mouse pointer to the inner left corner of the upper horizontal exterior wall.
   - When the ***endpoint*** notation displays ***drag*** horizontally toward the center, releasing the mouse button anywhere.

     A *stroke construction* line is displayed through the inner edge of the upper exterior wall.

- Select the **Parallel Lines** tool.

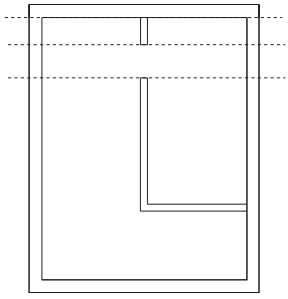
- ***Drag*** the construction line about **1 m** towards the middle of the room.
- Enter **1** for the delta distance in the status line and press **Enter**.

  The parallel line is drawn 1 m below the upper exterior wall
- ***Drag*** the new construction line about 1 m towards the middle of the room.
- Enter **1** for the delta distance in the status line and press **Enter**.

  The second parallel line is drawn 1 m below the second construction line.
- Select both parallel construction lines with the **Selection** tool.
- Select the **Simple Trim** tool from the tool palette.

- Click the **Simple Trim** tool on the vertical interior wall between the two construction lines.

  The interior wall is trimmed and the wall segment between the two construction lines is deleted.



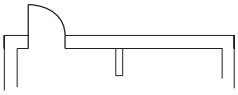
8. Add a *1 m* door beginning in the upper horizontal exterior wall and a *2,5 m* window in the right wall. Both should *25 cm* from the wall corner.
   - Activate the Layer **Exterior Walls** in the Layer indicator.
   - Choose the **Symbolmanager** command in the **Symbols** submenu from the **File** menu.

     The **Symbolmanager** is displayed.

   - Choose in the **Symbolmanager** the library **A Door top**.
   - Select the symbol **Single Door with one leaf**.
   - Select from the Part List **Door 1,01/24**.
   - Enter into the field **L** the distance from the wall corner **0,25**.

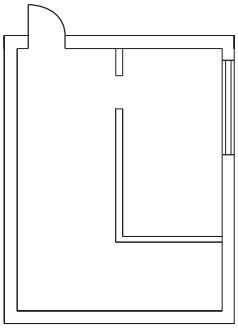
     Press the mouse button at the corner of the upper inside wall and ***drag*** horizontally toward the center, releasing the mouse button anywhere.

- Click the **Set** button.



The wall geometry is redrawn to accommodate the door.

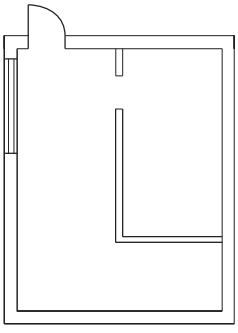
- Select the library **A Windows top**.
- Select the symbol **Simple Window**.
- Select from the Part List **Window 2,51/24**.
- Enter into the field **L** the distance from the wall corner **0,25**.
  Press the mouse button at the corner of the upper inside wall and ***drag*** vertically toward the center, releasing the mouse button anywhere.
- Click the **Set** button.



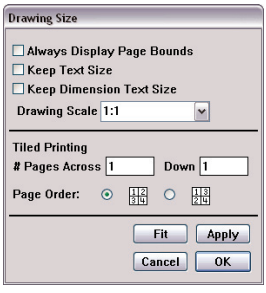
The ***drag*** indicates the orientation of the door as well as which way it opens

The wall geometry is redrawn to accommodate the window.

9. Move the window and observe how the walls are automatically reconstructed.
   - Select the window.
   - ***Drag*** it to the opposite wall.



10. Set the paper size and drawing scale.
    - Choose **Print Setup** from the **File** menu.
    - Select paper size **DIN A4** and click **OK**.
    - Choose **Drawing Size** from the **Layout** menu.
      The **Drawing Size** dialog box displays.



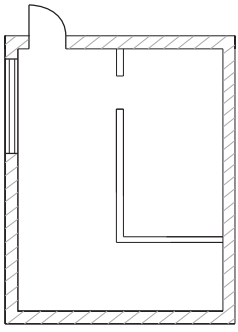
The available paper sizes depend on the printer or plotter you have specified.

- In the **Drawing Scale** entry box, type **1cm:50cm** (this entry is the same as **1:50**)
- Click **OK**.

The **Scale** entry box accepts combinations of units of measure.

If you want to crosshatch smart walls containing smart symbols you have to segment the walls first to create hatch boundaries. Since smart symbols cover only wall segments they can't be used as hatch boundaries.

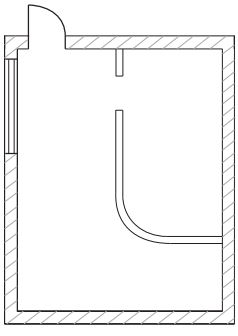
11. Segment the exterior walls.
    - Select the two symbols (door and window).
    - Choose the **Segment** tool from the tool palette.
    - Click with the **Segment** tool the wall segments besides the door and the window symbol.
      The walls are segmented.
12. Crosshatch the exterior walls.
    - Select by clicking with the **Selection** tool all visible wall segments of the exterior walls.
    - Choose **Hatch** from the **Pen** menu.
      The exterior walls are crosshatched.



13. Ungroup the interior walls.
    - Select the lower vertical and the horizontal interior wall.
    - Choose **Ungroup** from the **Arrange** menu.
      Both wall segments are ungrouped and loose their smart features.
14. Fillet the corner of the two interior wall segments.
    - Click the **2-Entity Fillet** tool.

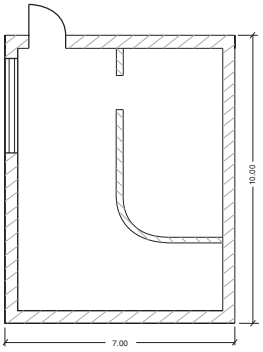
You can fillet or chamfer smart walls only if you first ungroup them. Since smart walls will loose their smart features by ungrouping them, you should place all symbols before you fillet any smart wall.

    - Enter in the status line **0,4** for the radius of the fillet.
    - Click with the **2-Entity Fillet** tool the two ungrouped interior walls, to fillet the inside and outside corners.
    - Enter in the status line **0,6** for the radius of the fillet and press **Enter**.
      The interior walls are filleted.



15. Crosshatch the interior walls.
    - Select the upper vertical interior wall segment and all lines building the filleted interior walls.
    - Choose **Crosshatch** from the **Pen** menu.
      The **Crosshatch** dialog box appears.
    - Enter **-45°** for the hatch angle.
    - Click **OK**.
      The interior walls are crosshatched.

16. Dimension the sides of the building.



17. Save or discard the file, as you wish.

Accomplishments

- Using the Smart Wall tool
- Using smart window and door symbols
- Filleting walls.
- Crosshatching walls
- Scaling a drawing

Appendix

Operators and Units
Special Characters
Data Interfaces
Plotter Fonts
Crosshatch Patterns
DraftBoard.ini
Customer Services

Operators and Units

This appendix describes mathematical and conditional operators.

Mathematical Operators

Many entry boxes accept mathematic, trigonometric, and exponential operators in the form:

funcName (arg1; arg2; . . . ; argN)

Operator	Example	
Addition	3+0,5	= 3,5
Subtraktion	3-0,5	= 2,5
Multiplikation	3*0,5	= 1,5
Division	3/0,5	= 6
Square Root	sqrt(2)	= 1,4142...
Parenthetical	3/(4+(1/0,5))	= 0,5
Scientific Notation	4e-3	= 0,004
Exponentiation	exp(2;7)	= 128
Sine of Angle	sin(30)	= 0,5
Cosine of Angle	cos(60)	= 0,5
Arctangent	atan(1;1)	= 45 (°)
Log (Base 10)	log(100)	= 2
Natural Log (Base e)	ln(2)	= 0,6931
Remove Fractional Part	truncate(1,4142)	= 1
Absolute Value	abs(-2)	= 2
Smallest Larger Integer	ceiling(1,4142)	= 2
Largest Smaller Integer	floor(1,4142)	= 1
Negative Value	neg(2)	= -2
Round	round(1,41)	= 1
Random Number	rand()	= (random number)

For divisions also a colon may be used instead of a slash.

Math operators are case-sensitive; so be sure to type them in lower case when you use them. You can use these operators to edit objects, make entries in the status line, and specify values in dialog boxes.

Units of Measure

You can use different units of measure such as inches (**"**), feet (**'**), inches and feet (**x'y"**), millimeters (**mm**), centimeters (**cm**), and meters (**m**). You can also mix the units in the mathematical expression as long as you label the units properly—for example, **10" + 25.4 cm**.

Decimal Indicators

DraftBoard can handle both *American* and *European* decimal entries in the status line (**24,5** or **24.5**).

A

Special Characters

You can use special characters and accents which are available with your computer. Usually these characters are described in the user manual of your *Operating System*.

Special Characters for Microsoft Windows

You can enter many *special characters* from the keyboard. If necessary, unlock the keypad with the **Num Lock** key, and then hold down the **Alt** key and enter the numeric code for the character you want.

The **ANSI**-Code you enter must have **4** digits. If the code exists only of **3** digits you must place a leading **0**, since the *four digit* code **0216** doesn't have the same result as **216**.

The character appears in the current font (including ***DraftBoard's*** Plotter font). The codes are listed in the *Microsoft Windows User's Guide*.

You can as well use the *Windows* ***Charset*** in the **Accessories** submenu of the **Start menu** to insert special characters into ***DraftBoard***.

Examples:	°	=	**Alt 0176**
	Ø	=	**Alt 0216**
	ø	=	**Alt 0248**
	±	=	**Alt 0177**
	©	=	**Alt 0169**
	@	=	**Alt 64**
	€	=	**Alt 0128**

Data Interfaces

This appendix contains additional information on the data interfaces already described in the manual.

DWG/DXF Interfaces

In general, it is recommended that you show all your layers before you export from ***DraftBoard*** or any other CAD package so that you know what objects are exporting.

- ***DraftBoard*** will always export only the current model. If your file contains *Detail Views*, flatten them with the **Flatten View** command found in the **Views** menu. If you have dimensions already in the detail view and the view scale is set to **1**, you can group the dimensions and geometry together before flattening the view. This will bring the dimensions across to the current model for Export.
- ***AutoCAD*** doesn't accept any **&** character in a layer name. Rename therefore all layers having a **&** character in their name. Otherwise AutoCAD will not import the file and generate an error message instead.
- In case line patterns are disorted (too big or too small) in ***AutoCAD*** after importing a ***DraftBoard DXF*** *File*, change the ***AutoCAD*** variable **LTSCALE** to a smaller number to display the line patterns at an appropriate scale.
- In case that after importing a ***DraftBoard DXF*** *File* into **AutoCAD** the arrowheads of some angular dimensions are displayed in ***AutoCAD*** in the wrong direction (pointing towards the text rather than towards the extension lines), execute the **DIM** command in ***AutoCAD***, select the affected dimensions, and use the **UPDATE** command to display the arrowheads.

DXF Configuration Files

The two configurations files ***DXF****IN.****CFG*** (Import) and ***DXF****OUT.****CFG*** (Export) allow you to set additional parameters for *colors*, *line patterns*, *line weights* and *fonts*. These two configurations files work like filters considering all parameters specified in these files when importing or exporting ***DXF****- and* ***DWG****-Files*. Both configuration files must be in the **DraftBoard** folder. Before modifying any of these files you should read the related *readme files* ***DXF****IN.****TXT*** and ***DXF****OUT.****TXT***. Both files are ***ASCII*** *Files* that can be modified using any simple *Text Editor*.

How exporting affects some objects

It is difficult to exchange data between different graphics programs because each program defines geometric objects differently. Even though the **DXF/DWG** and **IGES** formats standardize this exchange, some objects do not transfer exactly. The following table tells you what to expect when you export ***DraftBoard*** objects.

DraftBoard	DXF/DWG
Hatches	Hatches are exported as groups, except the option **Ungroup hatches** is active, then they are converted in normal lines.
Detail views	All detail view will be exported, as long as not more than two models are displayed in ***DraftBoard***. In addition the selected ***AutoCAD*** *Version* must support **Modelspace** and **Paperspace** (*Version* ***11*** or higher). If not all *detail views* must be flattened.
Ellipses	Ellipses are exported into ***AutoCAD*** *Version* ***13*** or higher. For all previous versions ellipses are approximated by line segments.
Fonts	Fonts that are defined in the ***DXF*** *Configura-*

The *Sheet View* will be exported into the AutoCAD ***Paperspace*** and *all Detail Views* into the *AutoCAD* ***Modelspace***.

	tion files are maintained.
Layer	Layer will be maintained.
Locked Objects	Objects are transferred correctly, but unlocked.
NURB-Splines	***NURB** Splines* are exported using ***AutoCAD** Version **13*** or higher. For all previous versions ***NURB**-Splines* are approximated by line segments.
Smart Walls	Smart walls will be represented by simple lines.
Text	Multi-line text in a text frame is exported as separate text lines.
Tolerances	*Tolerances* are exported using ***AutoCAD** Version **13*** or higher. In all previous versions tolerances will go lost, but the dimensions are maintained.

DraftBoard	IGES
Crosshatching	The crosshatching is converted to symbol representation.
Detail views	The view is lost but the dimensions and text are retained.
Diameter dimension	The leader style may change.
Fonts	Fonts are not maintained.
Grouped geometry	The geometry is retained but it is no longer grouped.
Layers	The layers are lost but the geometry is retained.
Locked geometry	The geometry is retained but it is no longer locked.
Radial dimensions	The leader style may change.
Smart walls	The double lines are converted to symbol representation.
Text	A multiple line note becomes many single lines.
NURBs	***NURB** Splines* are supported.

Plotter Fonts

You can use with ***DraftBoard*** all ***TrueType*** and ***PostScript Fonts*** available on your computer. In addition ***DraftBoard*** offers own *Plotter Fonts* that are listed below:

Plotter

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöüˆ°!”§$%&/()=?'+'−*/#'_.,:;<>
1234567890

Plotter Extended

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöü↑°!''§ $%&/()=?'+'−*/#'_.,:;<>
1234567890

Plotter Greek

ΑΒΧΔΕΦΓΗΙϑΚΛΜΝΟΠΘΡΣΤΥ∇ΩΞΨΖ
αβχδεφγηιϑκλμνοπρστυ∈ωξψζ
ˆ°!”$%&/()=?'+'−*/#'_.,:;<>
1234567890

Plotter Roman

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnoprstuvwxyz
ÄÖÜäöüˆ°!”§$%&/()=?'+'−*/#'_.,:;<>
1234567890

DraftBoard contains in the **Fonts** subdirectory in the **DraftBoard** folder sample files that show for each character the related ***ANSI*** *Code.*

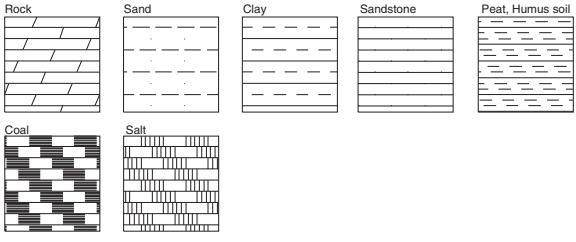
In addition to the plotter fonts shipping with ***DraftBoard*** you can use all plotter fonts with the ***.shx*** *Format. You* just must copy these ***.shx*** files into the **Fonts** subdirectory of the **DraftBoard** folder. See as well in the *Text* chapter of this manual.

A

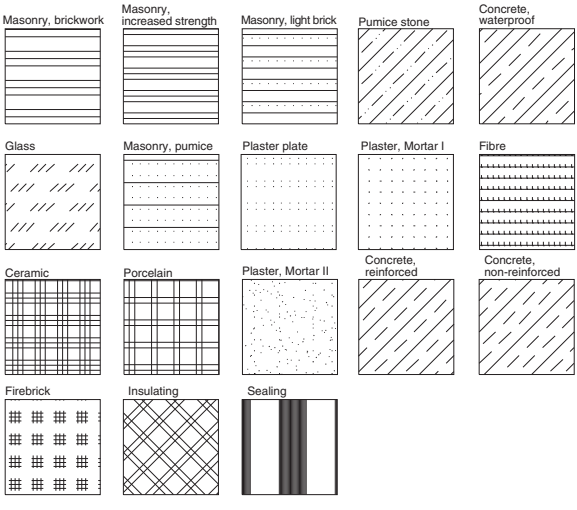
Crosshatch Patterns

DraftBoard offers a multitude of associative *DIN* and *ANSI hatch patterns*, of which you can modify the *spacing* and *angle*. You will find these hatches organized in groups such as *metals*, *fluids* etc. in the **Hatch** dialog box. All hatch patterns contained in this library are listed below.

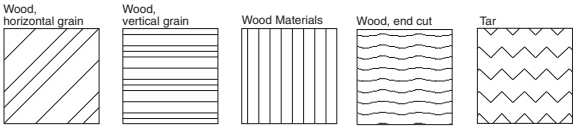
Mineral Materials



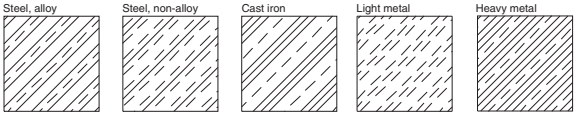
Other natural Materials



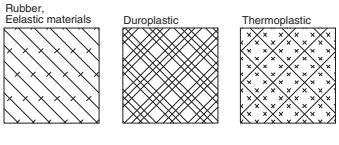
Natural Plant Materials



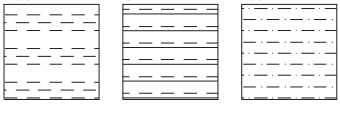
Metal



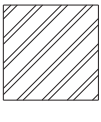
Synthetic Material



Fluids



Solids



A

DraftBoard.ini

The **DraftBoard.ini** file contains all default settings ***DraftBoard*** uses for any new drawing. The **DraftBoard.ini** file is an ***ASCII*** *text file* that can be opened and edited with any text editor.

To make changes to the **DraftBoard.ini** file open it from any text editor and make the desired changes. Save the file as a text file. The next time ***DraftBoard*** is launched it will take account for the new setting made in the **DraftBoard.ini** file.

The **DraftBoard.ini** file contains the following sections:

[User]

This section contains the *Name* of the registered user and the registered company

Name=
Company=

Both names will be automatically entered when installing ***DraftBoard***.

[Defaults]

In this section you can enter the *Path* and *Name* for the ***DraftBoard*** *Preferences file* **Prefs.vlm** as follows:

PrefFile=c:\Programs\DraftBoard\prefs.vlm

You only should specify a path for the *Preferences file* in case you don't want to use the *Preferences file* in the **DraftBoard** folder.

Creating a individual Preference files for a Network version

When you want to create an individual *Preference file* for each user of a ***DraftBoard*** *network version* you proceed as follows:

1. Create on the *Network Server* a folder (for example named **Preference**) for all *Preference files*.
2. Create folders for each user in this folder.
3. Copy the files **DraftBoard.ini** and **Prefs.vlm** in each folder.
4. Specify in each **DraftBoard.ini** file in the **Default** section the *Path* for the *Preference file* for the specific user:

   PrefFile=d:\Preferences\Smith\prefs.vlm

   where **Preferences** in this example is the name of the general *Preferences* folder and **Smith** the folder name of one user.
5. Select the ***DraftBoard*** *Program Icon* on the Windows Desktop.
6. Press the right mouse button and select **Properties** in the displayed menu.
7. Enter in the **Target** entry box behind **DraftBoard.exe** the path for the **DraftBoard.ini** file for the respective user as follows:

   C:\Programs\DraftBoard\DraftBoard.exe /
   i:d:\Preferences\Smith\DraftBoard.ini

;FILE LOCKING

FileLock=ON

When **File Locking** is activated (**= on**) all users can access a file in a network, but only the user who opened the file first has the right to save this file. For a single user license **File Locking** should be deactivated (**=off**).

File Locking works only if there exists a file named **public.key** on the *File Server* on the network. This Public.key file can be created with any text editor. The **Public.key** file can be even empty, it should be only placed in the main directory of the *File Server*.

;DISPLAYED FONTS

This section let you specify the number of fonts that will be displayed in the **Font** submenu from the **Text** menu. Fonts that do not show up in this menu can be accessed in the **Other** dialog box.

If you specify

MaxFonts=30

a maximum number of 30 fonts will be displayed.

;SCREEN

This option let you set the size of the screen that ***DraftBoard*** opens to.

Screen=FULL

opens to a maximized window.

Screen=Standard

opens to the standard size, which is almost maximized, but with room for icons at the bottom. If no value is specified, the screens opens to the standard size.

[RecentFiles]

The maximum number of files that can be displayed in the *Recent File List* at the end of the **File** menu is **8**.

Files=6

[Symbol]

This section contains the path of the **Symbol** folder relative to the **DraftBoard** folder

VStandard=Symbols

All further paths specified describe the absolute paths for the symbol palettes in case you added them to the **Utility** menu using the **Add command** in the **Utility** menu.

VMech=C:\Programs\DraftBoard\Symbols\Engineering
VArch=C:\Programs\DraftBoard\Symbols\Architecture

[SymTab]

This section contains the position of the **Parameter** dialog box that you can open in the **Symbols** dialog box.

LISTBOX=[80 140 356 358]

[Palettes]

This section contains all palettes available in ***DraftBoard***. Behind each palette name the position of the palette is specified and in a second section if the palette is visible or not (**true/False**):

Misc=190 80
Windows=100 275
Drawing=130 275
Bom=30 350
Dimension=10 345
Functions=30 440

MiscVisible=false
WindowsVisible=false
DrawingVisible=false
BomVisible=true
DimensionVisible=false
FunctionsVisible=false

Customer Services

Thank You for joining the world of ***DraftBoard***.

Registration

By registering your new software, you'll receive:

- The right to *Updates* and *Upgrades*,
- Replacement of damaged media,
- Notification of product *updates*, *upgrade* and *new products*,

Filling out and returning the registration card is a painless process. Don't forget to enter the name of the respective user, even if the software is registered to a company.

Media Replacement

If your product is defective or damaged, all defective or damaged media will be replaced at no cost during the 3 month warranty period.

Updates and Upgrades

As registered user you will be notified of product *updates*, *upgrade* and *new products*,

- **Updates**: When we make minor improvements to a product, we will release an updated version.
- **Upgrades**: As we continually drive our technology forward, we may develop significant new features that enhance our existing offering. Such developments are incorporated into a new version of the software.

Product Support

All registered ***DraftBoard****-Customers* may call directly POSH GmbH for any support issues.

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63897 Miltenberg

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Telefax: +49 (0) 93 71 - 69 241

E-Mail: support@draftboard.de

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