KeyCAD™ Deluxe 2D
Version 3.0
User's Guide

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Getting Started

Welcome to *KeyCAD™ Deluxe 3.0*, the most accurate, advanced and easy-to-use *KeyCAD* ever. CAD, or Computer Aided Design, lets you create geometrically precise drawings quickly and accurately. Like previous versions of *KeyCAD*, *KeyCAD Deluxe* is designed for value, speed, and ease of use. With its wealth of new features and options, *KeyCAD Deluxe* is suited to the most demanding professional 2D *and* 3D CAD applications. The front section of this user's manual discusses the 2D application. The complete 3D version is in the back of this manual.

This chapter covers how to get the information you need to learn and use *KeyCAD*.

NOTE: *If you are a KeyCAD for Windows 2.0 user, read the Appendix "Tips for KeyCAD 2.0 Users" at the end of this manual.*

System Requirements

To run *KeyCAD*, your computer system needs to meet the following minimum requirements:

- 386 or better CPU

- 8MB recommended

- 11MB of free hard drive space

- VGA graphics card or better

- Windows 3.1 or greater

NOTE: *KeyCAD comes in two versions, one for Windows 3.1 and one for Windows 95. The Windows 95 version is fully Windows 95 compliant, and both versions comply with the Microsoft Office standard.*

Installing *KeyCAD*

KeyCAD's installation procedure varies slightly, depending on whether you are running Windows 3.1 or Window 95. Before installing *KeyCAD*, please exit all programs that are running on your system, except for Program Manager.

To install *KeyCAD™ Deluxe:*

1.   Insert the appropriate *KeyCAD* CD-ROM into your CD-ROM drive.

2.   If you are using Windows 3.1, choose Run from the File menu in the Windows Program Manager. If you are using Windows 95 or higher, click Start, then choose Run from the popup menu.

3.   Type **D:\SETUP** in the dialog Run field (or substitute for **D** the letter of the CD-ROM drive you are using), then press <Enter>.

4.   Follow the onscreen instructions to complete the installation.

If at any time you want to remove *KeyCAD* and all related files from your system, double-click the **Uninstall** icon and follow the onscreen instructions.

Registration

To receive free technical support, as well as information on new product releases, please register your copy of *KeyCAD* today. Just fill in and mail your registration card, or call us at (423) 670-2020.

Learning to Use *KeyCAD*

KeyCAD comes with a combination of printed and online documentation that can help you learn to use the program and provides answers to your questions as you delve deeper into *KeyCAD's* many features. If you are new to CAD programs, we recommend that you start using the documentation as follows:

1.   Read Chapter 2, *Quick Tour* (in the *KeyCAD* user's manual) and work through the "mini-tutorial".

2.   Choose Tutorial from the *KeyCAD* Help menu. The tutorial window will remain open on your screen while you work through the exercises.

3.	After you become familiar with *KeyCAD's* basic features, read the user's manual for more in-depth information, and use the online Help system for a quick lookup of *KeyCAD's* features while you are drawing.

Running the Tutorial

KeyCAD's electronic tutorial provides step-by-step, hands-on instructions that teach you how to create drawings in *KeyCAD*. The tutorial remains on top of your *KeyCAD* window so that you can draw in *KeyCAD* while following the instructions.

To start the tutorial, choose Tutorial from the Help menu. The Tutorial Topics page will appear. You can go to any tutorial topic by clicking on the topic. Click on the buttons at the top of the tutorial screen to navigate.

At the bottom of many of the tutorial topics, you will find a "Show Me" button. You can click on this button at any time during the tutorial to see what your drawing should look like.

Using the Manual

The *KeyCAD* user's manual has been designed to help users of all levels learn *KeyCAD,* and provide quick and easy reference to all *KeyCAD's* features. This section explains the organization of the manual, tells you how to quickly access the information that you need, and describes the manual's typographic and terminological conventions.

How the Manual is Organized

The manual is divided into two parts; the first half contains instructions for the *KeyCAD 2D* program. The second half describes *KeyCAD 3D*.

The first two chapters of this manual, *Getting Started* and *Quick Tour*, are designed to get you up and running with *KeyCAD,* and introduce you to its interface and basic features.

Each of the remaining chapters is a comprehensive reference to a set of *KeyCAD* features. The first half of each reference chapter discusses *KeyCAD* concepts and techniques; the second half is a Command Reference to all of the commands and dialogs available from one of *KeyCAD's* menus.

Cross References

Cross-references to related topics are contained within "See Also" paragraphs at the end of the section to which they apply:

SEE ALSO: *List of related topics*

The index includes all of the topics listed in these See Also lists, so that you can easily locate the exact page number of the cross-reference.

Reference Headings

Commands and tools that can be accessed from a menu or toolbar are described in *reference headings* that incorporate information on all possible ways to access the command.

Typographic Conventions

This user's manual uses the following typographic conventions:

- The names of keyboard strokes are enclosed in angle brackets. A plus (+) sign indicates key combinations. For example, <Ctrl>+<Y> means to press the Y key while holding down the Control key.

- Menu items are referred to by a *menu path*, in which the steps through the menu system are separated by a pipe (|). For example, File|Open refers to the Open menu item, located on the File menu. If there is a submenu off of the menu item, then there will be one more pipe. For example, Modes|Snaps|Vertex means the Vertex command in the Snaps submenu of the Modes menu.

- Windows dialogs can contain multiple sheets accessible by clicking on tabs. This manual will refer to these sheets as *property sheets*. When referring to a control located in a dialog, the manual will also refer to the specific property sheet where the control is located.

- User entries are shown in boldface type. For example, you might see instructions like this: "Type **1.25 ft** into the Length field on the edit bar."

Terminological Conventions

At times, discussions involving closely related but distinct concepts can become confusing. It will help if you keep the following terminological distinctions in mind.

Commands and Tools

The terms *command* and *tool* are used in the manual to describe distinct types of *KeyCAD* functions.

Command. Commands are *KeyCAD* functions that take some action and complete the action. Select All would be an example of a command, because it immediately selects all of the objects on the screen and returns

KeyCAD to its previous state, expecting no further action on the part of the user.

Tool. When you select a tool, you access a special set of functions that are available as long as the tool is *active*. Some tools remain active until you explicitly switch to another tool. For example, you can activate the Single Line tool and continue to draw lines until you switch to another tool. Other tools, such as Zoom Window, become automatically inactive as soon as you finish using them.

TIP: *Think of a* command *like this—you tell KeyCAD what to do, and KeyCAD does it. Think of a* tool *like this—you ask KeyCAD for the means to perform an action, KeyCAD provides that means, then you perform the action.*

NOTE: *Occasionally, the term* tool *can also be used to simply mean a tool button on the toolbar, or any function accessible using a tool button.*

Objects and Entities

The term *object* is a general term used in this manual to mean anything that can be displayed in a *KeyCAD* drawing. The term *entity* is used in a much more narrow sense, meaning a drawing object created with *KeyCAD* drawing tools. Entities include single lines, multilines, arcs, circles, polygons, and so forth, as well as construction lines and circles. Entities can be combined together into groups, which are also referred to as objects. Other types of *KeyCAD* objects include bitmaps, OLE objects, and symbols.

Using the Help System

KeyCAD's online help system provides a complete reference to *KeyCAD's* features, including information on *KeyCAD* scripting that is not covered in this manual. To access the help system, choose *KeyCAD* Help Topics from the Help menu. To learn more about the features of the Windows help system, choose Using the Help System from the list of topics.

Context-Sensitive Help

If you need particular information about a tool, command, or dialog, you can go directly to the relevant help topic from the *KeyCAD* interface. To view the help topic on a tool or menu item, just place the mouse cursor over the tool, or highlight the menu item, then press <F1>. Or, you can click on the Help icon, then click on the tool you want information about.

Contacting Technical Support

If you can't find the answer to a question in this manual or the Help system, The Learning Company Technical Support staff is there to help you. You must be a registered *KeyCAD* user to get technical support.

Before you call, have your computer turned on. If you have a question about how something in *KeyCAD* works, then you only need to have the first three of the following items available. If you are having a system-related problem, then please try to have all the following information available:

- Your name, address, and phone number.
- Product name and version number (choose Help|About).
- Computer make and model.
- The version of Windows that you are running.
- Amount of RAM installed and type of memory manager.
- Type and capacity of your hard disk.
- Graphics adapter and monitor type.
- Printer/plotter make and model.
- List of any special peripherals installed.
- The text of the error message and a concise and clear description of the problem.

You can contact Technical Support at the following numbers:

Direct Telephone	1-423-670-2020
Fax	1-423-670-2021
Internet	support@learningco.com
Compuserve	GO LEARNINGCO
WWW	http://www.learningco.com
Correspondence	The Learning Company, Inc.
	Technical Support Center
	9715 Parkside Dr.
	Knoxville, TN 37922

Customer Service

For replacement CDs or information about other products, contact our Customer Service Department at **617-494-5700**.

For more information about The Learning Company, visit our World Wide Web site at: **http://www.learningco.com**.

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Quick Tour

This chapter will take you on a quick tour through *KeyCAD's* basic features, so you can start using the program right away. It will cover the following topics:

- Starting *KeyCAD*
- The *KeyCAD* desktop
- Simple drawing
- Using the *KeyCAD* interface
- Creating and printing a drawing
- Using Symbol Libraries

After you've finished this chapter, if you'd like to work through more tutorials, just choose Tutorial from the *KeyCAD* Help menu.

Starting *KeyCAD*

If you haven't already installed *KeyCAD*, install it following the instructions in the previous chapter. The way you start *KeyCAD* differs, depending on whether you are running Windows 3.1 or Windows 95.

Starting *KeyCAD* in Windows 3.1

When you install *KeyCAD* in Windows 3.1, a *KeyCAD* icon is placed in a program group called Creative Office. To start *KeyCAD* from Windows Program manager:

1.   If the Creative Office program group is iconized, double-click the Creative Office icon to open the program group.

2.   Double-click the *KeyCAD* icon.

Starting *KeyCAD* in Windows 95

When you install *KeyCAD* in Windows 95, *KeyCAD* is installed in a *KeyCAD* folder and placed on the Windows Start menu. To start *KeyCAD*:

1. Click the Start button, then choose Programs to open the Programs submenu.

2. Click Creative Office to open the Creative Office submenu.

3. Click *KeyCAD* 3.0 on the Creative Office submenu.

Choosing a Template

After *KeyCAD* starts, the first thing you will see is the Tip of the Day dialog. Click OK when you've finished reading the Tip. *KeyCAD* will then display the Create From Template dialog.

Create From Template   ☒

Template:

| Ansia_a |
| Dboard |
| Dbrdmetr |
| Elecmetr |
| Electri1 |
| Electric |
| Iso_a4 |
| Ldscpln |
| Ldspmtr |
| Normal |
| Normmetr |
| Quiktour |
| Tutor1 |

OK
Cancel
Open...
Set Default
Help

Description
For use with manual Quick Tour.

☑ Show Preview
☐ Don't ask again
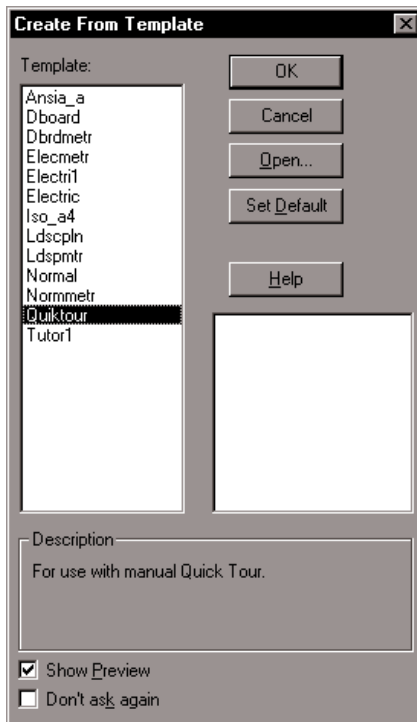
Choose the QuikTour template from this dialog

This is where you choose a standard template to use as the basis of your drawing. Click Quiktour in the box labelled Template, then click OK to close the dialog.

SEE ALSO: *Create from Template Dialog*

KeyCAD Desktop

Once you have chosen the Tutorial template you will see the *KeyCAD* desktop with a new, blank drawing. The following illustration shows the basic components of the *KeyCAD* desktop that appear on your screen.
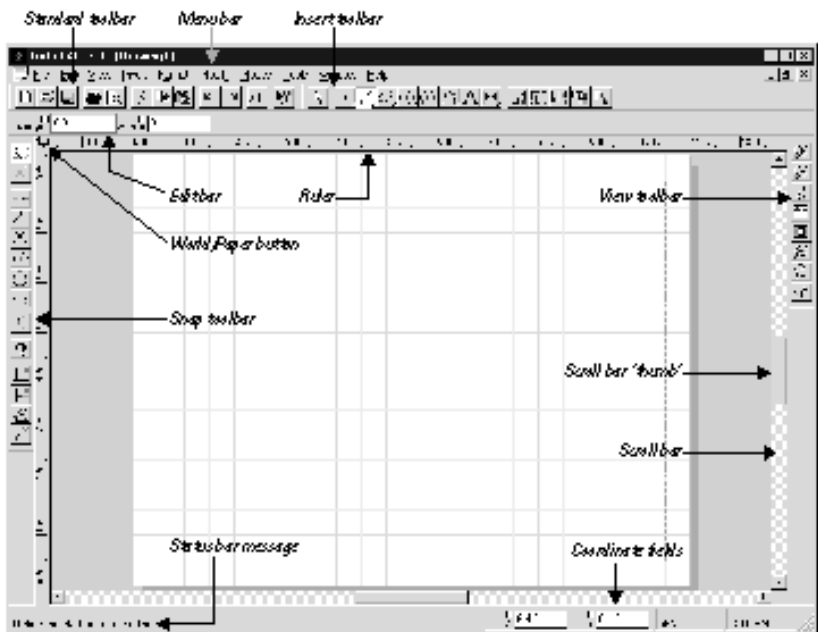
KeyCAD desktop

Here are some brief descriptions of the components on your screen:

Menu Bar. Click on the menu names on the menu bar to access *KeyCAD*'s commands.

Toolbars. Toolbars provide a convenient alternative to the menu system. Your screen is now displaying four toolbars: the Standard and Insert toolbars at the top of the screen, and the Snaps and View toolbars on the left and right sides of your screen, respectively. Clicking on any of the buttons on the toolbars will give a *KeyCAD* command.

TIP: *Notice that if you place the mouse cursor over a tool and leave it there briefly, a small yellow rectangle (or* tooltip*) will appear beneath the tool, displaying its name.*

Scroll bars. Your drawing can be larger (much larger!) than what you see on the screen. By clicking in the scroll bars, you can "scroll" to a different view of your drawing. Click on the arrow to scroll a small amount. Hold the mouse cursor over the "thumb" of the scroll bar (the rectangle in the middle of the bar) and drag it to move through your drawing. Click in the area between the thumb and the arrow to scroll a larger amount.

Paper. The Paper, the large white rectangle in the middle of the screen, shows you how your drawing will be layed out on the page when printed. You don't have to draw on the Paper; you can draw anywhere in the drawing space. The pattern of horizontal and vertical lines on the Paper is the grid, which marks exact locations in the drawing.

Ruler. The ruler gives you feedback about the size and position of objects in your drawing. Notice that the unit of measure of the ruler on your screen is feet. This is because the QuikTour template is set up to measure the objects you are drawing in feet. For example, you may be drawing a picture of a table that is five feet long, while the representation of the table is five inches long. Feet, in other words, are the drawing's *World Units*; inches are the drawings *Paper Units*.

SEE ALSO: *Units and Scale*

World/Paper Button. This button, in the upper-left corner of the ruler, lets you change the ruler so that it shows Paper Units. Click on the button, then observe how the ruler is changed. Notice that in this drawing one inch of paper is equal to one foot of the "real world" objects that you are going to draw. Click on the button again to change back to World Units.

Edit Bar. Use the Edit Bar to draw and edit using numeric values such as lengths and angles. For example, you could tell *KeyCAD* to draw a line exactly two feet long, at a 45 degree angle.

Coordinate Fields. Use the Coordinate Fields at the bottom of the screen to specify a point in the drawing space by its exact coordinates.

Status Bar message. This message area provides a brief message describing the currently active tool, or instructions about how to perform the next step in a drawing or editing process.

Simple Drawing

Let's start by drawing part of the some shapes just using the mouse.

1. Before you start drawing, make sure that all of your snap modes are turned off. Check the Snap toolbar on the left side of your screen and make sure that the No Snap button is highlighted.

2. Next, choose the Single Line tool from the Insert toolbar. (Or choose Insert|Line|Single from the menu.)

3. Now place the mouse in the drawing area and click. By clicking you have *defined the starting point* of the line. As you move the mouse around, you will see a *rubber band line* connecting your mouse to the starting point.

TIP: *Remember, if you are confused about what to do next in a command, read the status line at the bottom of your screen.*

4. Click the mouse again to finish the single line.

Now let's continue the exercise by drawing a circle.

1. Click the Circle Center and Point tool in the Insert toolbar.

2. Click anywhere in the drawing to define the center of the circle. As you move the mouse the circle stretches and shrinks.

3. Click once more to define a point on the circumference of the circle.

Congratulations, you have completed your first two drawing objects!
Your drawing may look something like this:
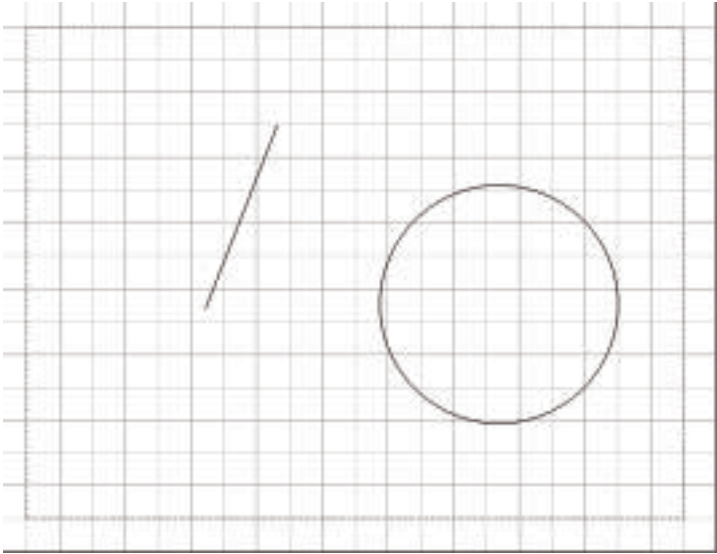
What to Do If You Make a Mistake

One of the important things to keep in mind as you begin drawing and
editing is that nothing is written in stone. *KeyCAD* allows you to undo
your previous drawing command, as well as several of the commands
that preceded it. To test this out, let's *undo* the line and circle you just
created, then *redo* them. We'll use the Undo and Redo buttons, located
on the Standard toolbar.

Undo Button   *Redo button*

1. Click the Undo button to undo the circle you just drew. The
   circle
   disappears.
2. Click the Undo button again to remove the single line.
3. Click the Redo button twice to restore the line and circle.
4. Choose Undo twice again to clear the screen for the next exercise.

TIP: *Remember that if you make a mistake or get confused, you can back up and move forward through a drawing to find a point from which you can continue.*

Drawing a Line with Snap Modes

Snap modes are a set of *KeyCAD* features that allow you to draw and edit with precision using the mouse. In this exercise we'll snap to the grid. (The grid is the pattern of intersecting blue lines that you see on the Paper in the center of the drawing area.) Follow these steps:

1. Start by choosing Snap to Grid, either by choosing Modes|Snaps|Grid or by clicking on the Snap to Grid tool on the Snap toolbar.

2. To get more visual feedback throughout this exercise, turn on the Magnetic Point tool. This lets you see the point where *KeyCAD* will snap to if you click the mouse.

3. Choose the Single Line tool from the Insert toolbar.

4. Place the mouse in near an intersection of two blue lines (that is, near a grid point), then click. The first point of the line "snaps" to the grid point.

5. Just for fun, place the mouse far away from any grid point (in the center of a square). When you click the mouse, *KeyCAD* beeps and refuses to do anything. This is what happens when you are too far from any possible snap points.

6. Now click the mouse near another grid point. *KeyCAD* snaps to grid point again, finishing the line.

NOTE: *All of the snap modes work essentially the same way. You can snap to the endpoints of lines, the middle of a line, the center of an arc, and so on. If you want to define a point at the location of the mouse click, use No Snap.*

See Also: *Snap Modes*

Deleting an Object

Now let's delete the line that you just drew:

1.   Choose the Select tool from the Insert menu.

2.   Click on the line that you just drew. The line should be enclosed in a selection rectangle:
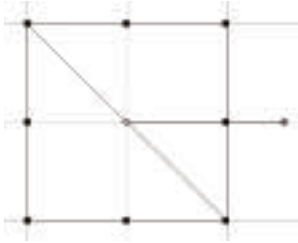
3.   Press the <Del> key: The line disappears.

TIP: *Once an object has been selected like this, you can delete it, move it, copy it, scale it, or rotate it using a wide variety of techniques. See the "Select Edit" section of Chapter 4 for details.*

Using the *KeyCAD* Interface

In this section you will explore how to use different components of the *KeyCAD* desktop to draw a single object. During this exercise, you will draw a *multiline*, which is a series of connected line segments. To draw the multiline you will use:

- •   a *flyout* toolbar
- •   the Coordinate Fields
- •   the Edit Bar
- •   the Snap toolbar
- •   the local menu

By using all of these tools together, you will get a glimpse of the many different ways you can approach a single drawing task.

During this exercise, you will practice moving between the Coordinate Fields, the Edit Bar, and the drawing area. Drawing will be easier if you remember these keystrokes:

- If you are in the drawing area or the coordinate fields, you can move the focus to the Edit Bar by pressing <Ctrl>+<E>.
- If you are in the Edit Bar or the coordinate fields, you can move to the drawing area by pressing <Ctrl>+<D>.
- If you are in the Edit Bar or the drawing area, you can return the focus to the coordinate fields by pressing <Ctrl>+<R>.

SEE ALSO: *Changing Focus*

Using Flyout Toolbars

You are now going to use a flyout toolbar to choose the Multiline drawing tool. Flyout toolbars are toolbars that "fly out" when you hold the mouse button down while the mouse cursor is over a tool button. Tool buttons that contain flyouts are marked by a small yellow rectangle in their lower-right corner. In this exercise use a flyout toolbar from the Insert toolbar.

1. Position the mouse cursor over the Single Line tool.

2. Hold down the left mouse button. Don't let the mouse button up until the flyout toolbar appears. The flyout looks like this:
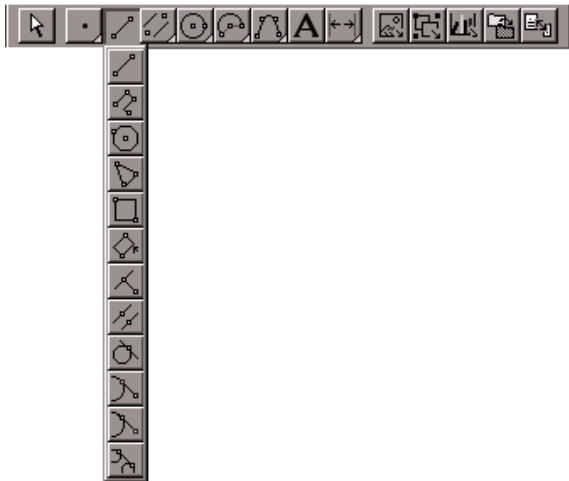
Flyout toolbar from the Single Line tool

3.   Click on the Multiline tool.

The Multiline tool is now *activated,* so that you can draw with it. It will remain active until you choose another tool.

NOTE: *Notice that the button on the Insert toolbar where the Single Line tool was now displays the Multiline tool.*

Using the Coordinate Fields

Let's begin by defining the first point of the multiline, using the Coordinate Fields. You'll do this by typing the exact coordinates of the point.

1. Press <Ctrl>+<R> to activate the Coordinate Fields, then type **1 ft** into the X box.

2. Press the <Tab> key to go to the Y Coordinate Field, and again type **1 ft**.

3. Press <Enter> to accept the contents of the fields. *KeyCAD* defines the starting point at precisely one foot on the X (horizontal) axis and one foot on the Y (vertical) axis. A small diamond marks this point.

Using the Edit Bar

You will now draw a line from the point you just defined by specifying the length and angle of the line in the Edit Bar. We want the line to be five feet long and point straight up.

1. Press <Ctrl>+<E> to move to the Edit Bar, then type **5 ft** into the Length field.

2. Press <Tab> to move to the Angle field, then type **90**. The Edit Bar should now look like this:

3. Press <Enter> to accept the contents of the Edit Bar fields.

KeyCAD draws a line five feet long at 90 degrees (straight up) from the first point you defined.

Using the Mouse and Local Menu

Let's draw a second multiline segment with the mouse and then finish the multiline using the local menu.

1. Check the Snap toolbar and make sure that the Snap to Grid tool is highlighted. (It should still be highlighted unless you changed it.)

2. Click on a grid point near the middle of the page. *KeyCAD* will add a new segment to the multiline.

3. Now click the right mouse button to open the local menu. Your screen should look something like this:
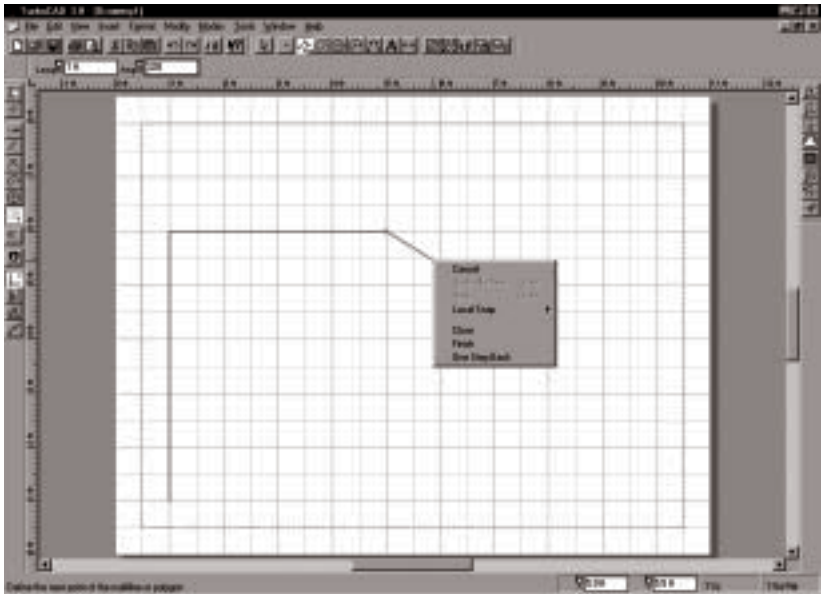
Using the local menu to finish the multiline

4. Click on Finish in the local menu. *KeyCAD* completes the multiline.

You have now drawn a multiline entity using *KeyCAD's* fundamental drawing controls: the Coordinate Fields, the Edit Bar, and the mouse in combination with the local menu. In the next exercise, you'll create a drawing using more of *KeyCAD's* drawing and editing tools, then save and print your drawing.

TIP: *Because the local menu is context-sensitive, it's not always obvious what functions it may contain in a given situation. While learning KeyCAD it's a good idea to check the contents of the local menu when you are using different tools and editing techniques. You may be surprised by what you find!*

SEE ALSO: *Editing with the Mouse and Local Menu, Finishing and Canceling, Local Snap, Snap Modes*

Creating and Printing a Drawing

In this exercise, you will create a simple pattern of circles and polygons. You will learn how to:

- Draw a circle using snap to grid.
- Change your view of the drawing by zooming.
- Inscribe a polygon in the circle using *local snap.*
- Select, move, and make multiple copies of objects.
- Save your drawing.
- Print your drawing.
- Exit *KeyCAD*.

NOTE: *Before starting the exercise, choose Edit|Clear|All to remove the previous drawing from your screen.*

Draw a Circle

1. From the Insert toolbar, choose the Center and Point Circle tool. (Or you can choose Insert|Circle|Center and Point from the menu.)

2. On the Snap toolbar, check to see that Magnetic Point and Snap to Grid are highlighted. If they are not, then click on them.

   Magnetic Point   **Snap to Grid**

3. Click near a grid point in the middle of the page, for example at the intersection of 4.5 ft on the vertical ruler (Y) and 5.5 ft on the horizontal ruler (X). This defines the center of the circle.

TIP: *To find out where your cursor is located, look at the markers that move along the horizontal and vertical rulers. You can also check the contents of the Coordinate Fields.*

4. Click on a point two feet (four grid points) immediately above the first point. The circle should now be finished and look like this.
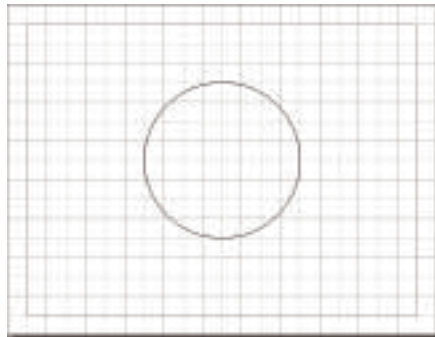
Zooming in on Your Drawing

KeyCAD's zoom and pan features make it easy to move to a specific view of your drawing. Right now let's use one of the most common zoom tools to move closer to the circle you just drew. Click the Zoom Extents tool on the View toolbar on the right side of your screen.

You can also hold down the <Ctrl> key and press the <Backspace> key, or choose View|Zoom|Extents from the menu. Your circle will now occupy the entire drawing space.

Inscribe a Polygon in the Circle

Now let's use snap modes to inscribe a polygon inside the circle using two more snap modes: Arc Center and Quadrant Point. This time, however, we'll use the local menu to set the snap modes. When you set a snap mode in the local menu, it is set temporarily, so it only lasts until you define the next point. This is called *local snap*.

To draw the polygon:

1. Position the mouse down over the line tool on the Insert toolbar (which should now be showing the Multiline tool, unless you've

changed it since the *KeyCAD* Interface exercise). Hold the left mouse button down, but **don't release it until the flyout toolbar appears.**

2. A flyout toolbar descends from the Insert toolbar. From this toolbar, choose the Polygon tool.

3. Right-click in the drawing area. From the local menu, choose Local Snap|Arc Center.

4. Click anywhere on the perimeter of the circle. Your first point should snap to the center of the circle.

5. Right-click again, and this time choose Local Snap|Quadrant Point from the local menu.

6. Click near the very top of the circle. Your point will snap to exactly 90 degrees at the top of the circle.

You have now inscribed a polygon precisely inside of a circle, using local snap. This illustrates how you can use local snap modes that you don't use very often (like Arc Center and Quadrant Point), to temporarily override a "basic" snap mode that you use most of the time (Snap to Grid). This is a very useful technique that you will find yourself using over and over again as you draw. Your drawing should now look like this:
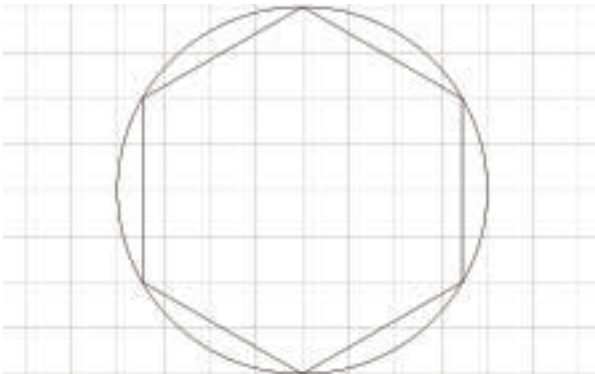
Selecting and Editing Entities

This section will illustrate how to use the Select Edit tools to select, move, and make linear copies of the objects you have drawn.

Selecting the Circle and Polygon

To select the circle and polygon:

1. Choose the Select tool from the Insert toolbar.

2. Click on the circle to select the circle.

3. Hold down the <Shift> key and click on the polygon to add it to the selection.

Moving the Selection

We'll now move the objects to a point in the lower-left portion of the screen so that we can copy them diagonally across the paper. (Make sure the Select tool and Snap to Grid are still on.)

1. Click the Full View tool on the View toolbar, so that the Paper fits in the screen.

(You can also zoom to Full View by pressing <Shift>+<Backspace> or by choosing View|Zoom|Full View fom the menu.)

2. Click on the yellow dot at the center of the selection (this is called the Reference Point). You'll know when the mouse is over this point, because the cursor turns into a four-directional pointer.

3. Move the mouse cursor to a point in the lower-left portion of the drawing, such as (2,2), and click on the grid point.

The selection will move, so that the Reference Point snaps to the grid point that you clicked on. Your drawing should look like the following:
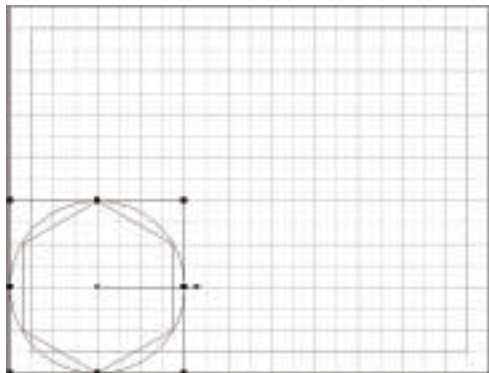
Making a Linear Copy

We'll now create multiple copies of the selection, aligned diagonally across the page:

1. Choose Edit|Copy Entities|Fit Linear. The Edit Bar should now look like this:

2. Click in the Sets field of the Edit Bar, delete the contents of the field, then type 20 into the field.

3. Click once in the drawing area to activate it.

4. Click on a grid point in the upper-right portion of the drawing, for example (8,6).

KeyCAD makes 20 sets (19 copies plus the original), aligned diagonally across the Paper.
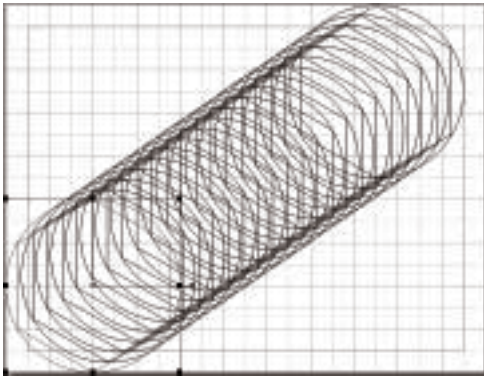
Saving Your Drawing

It's a good idea to save your drawing at this point. Click the Save button on the Standard toolbar (or choose File|Save from the menu).

Use the standard Windows dialog that appears to choose a directory in which you'd like to keep your drawing, and give it a file name. (One logical place to put it would be in the *KeyCAD* Drawings folder).

SEE ALSO: *Save As Dialog*

Printing Your Drawing

Before printing your drawing, let's have *KeyCAD* fit the drawing exactly on the Paper:

1. Choose File|Page Setup, then click the Fit button. The preview box now shows the drawing centered on the Paper.
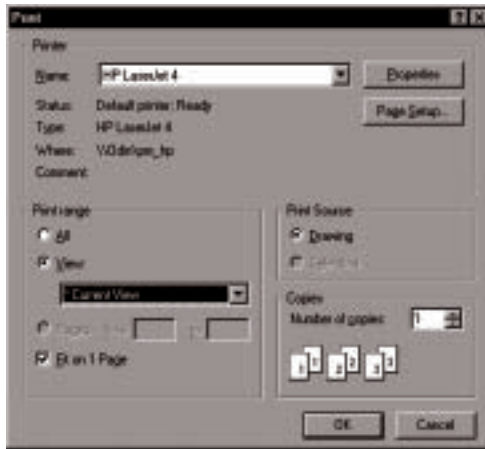
2. Click the Print button to print your drawing and close the Page Setup dialog.

3. Click the Save tool to save your drawing again, the choose File|Close to close this drawing.

Using Symbol Libraries

Symbol libraries make it easy to use thousands of standard drawing components available for use with *KeyCAD*. In *KeyCAD* 3.0, any Windows folder that contains *KeyCAD* drawings, or even drawings created in other programs that *KeyCAD* can read, can be used as a symbol library.

Let's start with a new drawing. Choose File|New from the menu, and choose the Quiktour template. Two symbol libraries have already been added to the Quiktour template for you. To see the libraries that have been added, choose Tools|Symbol Library from the menu. The Symbol Library property sheet will appear.

This property sheet is where you navigate to Windows folders that you want to use as symbol libraries. Notice that the Quiktour template is already attached to two symbol libraries, named *Kitchen* and *Bath*. If you wanted to add new libraries to your drawing, you would click the Add button and use a standard Windows dialog to navigate to a folder containing drawings that you want to use as symbols. See the "Symbol Libraries Property Sheet" section of Chapter 10 for more details. For now, click <OK> to close the dialog.

Once symbol libraries have been added to a drawing, you can view and access their symbols in the Symbols palette. To display the Symbols palette, choose View|Symbol Library.

Click on the Options button to display the Symbol palette's local menu. At the bottom of the local menu you will see the names of the two available libraries. Click on Bath to display the contents of the Bath symbol library.

Now let's place a symbol from the library in your drawing:

1. Position the mouse over any symbol, then hold down the mouse button.

2. Keeping the mouse button down, drag the symbol from the palette into your drawing, then release the mouse button. The symbol appears in your drawing.

To view the entire symbol, choose View|Extents (or press the View Extents tool button).

Exiting *KeyCAD*

If you like, save your drawing by choosing File|Save As, then choose File|Exit to exit *KeyCAD*.

For More Instructions...

This chapter introduces you to only a small portion of *KeyCAD's* many features. For a more detailed tutorial, choose Help|Tutorial from *KeyCAD's* menu and run the online tutorial. For comprehensive information on all of *KeyCAD's* features, read the rest of this manual.

3
File Management and Printing

File Menu and Dialogs

KeyCAD 3.0 provides all standard Windows features for creating, opening, and saving files. Its Open and Save features support several standard CAD file types. *KeyCAD's* printing features make it easy to print your drawings on single pages or to "tile" them across multiple pages that can be joined into a single large printout.

This chapter begins with a discussion of the file types supported by *KeyCAD*, followed by an overview of printing concepts and techniques. The remainder of the chapter provides a complete reference to the features accessible from the File menu.

NOTE: *The descriptions of dialogs in this chapter assume that you know the basics of working with dialogs in Microsoft Windows. If you need help with standard Windows interface techniques, please refer to your Windows documentation.*

Supported File Types

KeyCAD supports a variety of file types, including the standard .DXF and .DWG formats. This means that you can open files created in other CAD programs using a standard file format, such as .DXF, and then edit and save the files in *KeyCAD* format (.KCW); or you can save *KeyCAD* files as .DXF, and then edit and save them in another application.

KeyCAD also supports *KeyCAD 3D* .MDL files. You can save a drawing as an .MDL file in *KeyCAD* 3.0, open the file in *KeyCAD 3D*, and then use the features of *KeyCAD 3D* to add a third dimension to your drawing.

Or you can read an .MDL file in *KeyCAD* 3.0, if you want to edit a three-dimensional drawing using the more sophisticated drawing and editing tools available in *KeyCAD* 3.0.

The following table describes the file types supported by *KeyCAD* 3.0:

File Type	Extension	Comment
KeyCAD 3.0	.KCW	Default file type for *KeyCAD* 3.0.
KeyCAD 3D	.MDL	File type of *KeyCAD 3D*; *KeyCAD* 3.0 can open and save files of this type.
KeyCAD Template	.kCT	Use predefined or custom templates as basis for new drawings.
KeyCAD 2.0	.KEY	You can open, but not save, files as this type.
KeyCAD Symbols	.SLW	These files contain symbol libraries.
AutoCAD DXF	.DXF	Most common CAD file exchange type.
AutoCAD DWG	.DWG	AutoCAD drawing files.
HPGL	.PLT	A commonly used CAD file type (Hewlett Packard Graphic Language).

NOTE: *KeyCAD 3.0 can open and save KeyCAD 3D .MDL files. The appearance of the drawing files in the programs will vary slightly, because of the programs' different features and capabilities.*

WARNING: *If you open a file created in a previous version of KeyCAD, then save the file in KeyCAD 3.0, the file type will be updated to KeyCAD 3.0. You will not be able to re-open the files in the earlier version of KeyCAD.*

SEE ALSO: *Open Dialog, Save As Dialog*

Printing

KeyCAD 3.0 has a wide range of printing features, which make it easy to scale and center your drawing on the page, or to tile your drawing across multiple pages. *KeyCAD* also provides a full range of advanced printing options for experienced users.

NOTE: *Under Windows, plotters are treated as printers.*

Printer Paper and the Drawing Sheet

Before dealing with *KeyCAD's* printing capabilities, it will help if you understand a potentially confusing concept: printer paper versus the drawing sheet. *Printer paper* means the physical sheets of paper that go through the printer. The *drawing sheet* is the total surface area on which your drawing will be printed. The drawing sheet can correspond to a single sheet of printer paper or it can span multiple sheets of printer paper.

Both the printer paper and the drawing sheet have settable sizes, as well as landscape and portrait orientations. This provides a great deal of flexibility when you want to print a large drawing on multiple sheets. You could for example, print a poster in landscape orientation three feet wide and two feet tall. You could print this virtual sheet out on 32 sheets of 8.5" x 11" paper oriented in portrait, or on 30 sheets of the same size paper oriented in landscape.
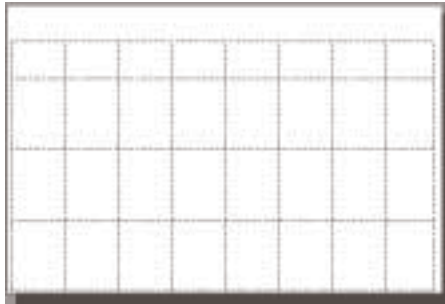
5' x 3' landscape drawing sheet on 8.5" x 11" portrait printer paper
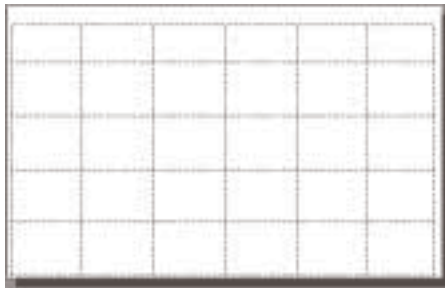
5' x 3' landscape drawing sheet on 8.5" x 11" landscape printer paper

Simple Printing

KeyCAD's many printing options may seem confusing at first glance, but in most cases you can print your drawing in just a few simple steps. Let's assume that you have created a drawing in World scale, without paying any attention to how your drawing will be set up on the page. Assume as well that you want to print on a single sheet of paper. Follow these steps:

1. Choose File|Print. The Print dialog will appear.
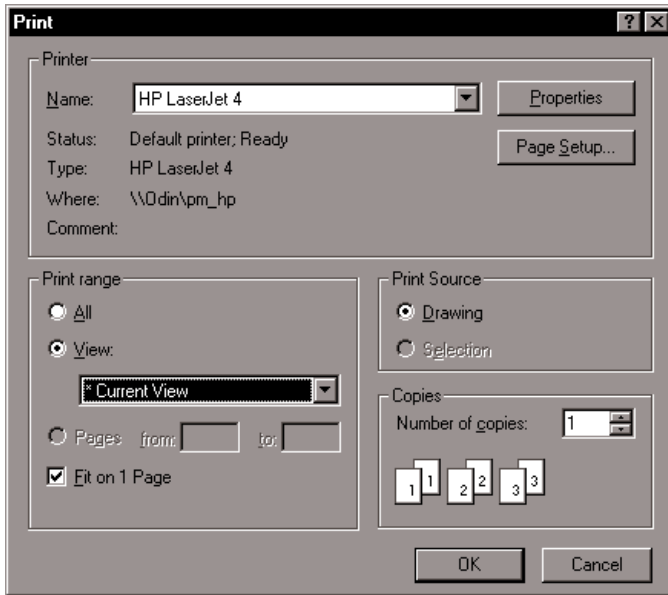
Print dialog

2. In the Print Range control group, click the All option to print your entire drawing.

3. Check the Fit on 1 Page option.

4. Choose a printer in the Name pulldown field, then click the Properties button to bring up the property sheets for your printer. Choose your paper size and paper orientation, the click OK to return to the Print dialog.

5. Click the OK button to print your drawing.

Your drawing will be centered and printed on a single sheet of paper.

SEE ALSO: *Print Dialog*

Tiled Printing

KeyCAD gives you two different approaches to printing a drawing on multiple tiled sheets of paper, using the Paper and Layout property pages of the Page Setup dialog. The two following sections describe these two options.

TIP: *When printing on multiple tiled sheets, it is usually best to have the Print Crop Marks option on, unless you set your margins to zero. Crop marks will enable you to easily trim the edges of the paper so that the sheets can be pasted onto a backing for display.*

SEE ALSO: *Page Setup Dialog, Print Dialog*

Tiled Printing by Setting Sheet Size

If you want to create a printout of a specific size tiled on multiple sheets, you can specify the size of the printout in the Paper property sheet. *KeyCAD* will determine the number and arrangement of the sheets of printer paper.

1. Choose File|Page Setup. The Paper property sheet will be visible in front.
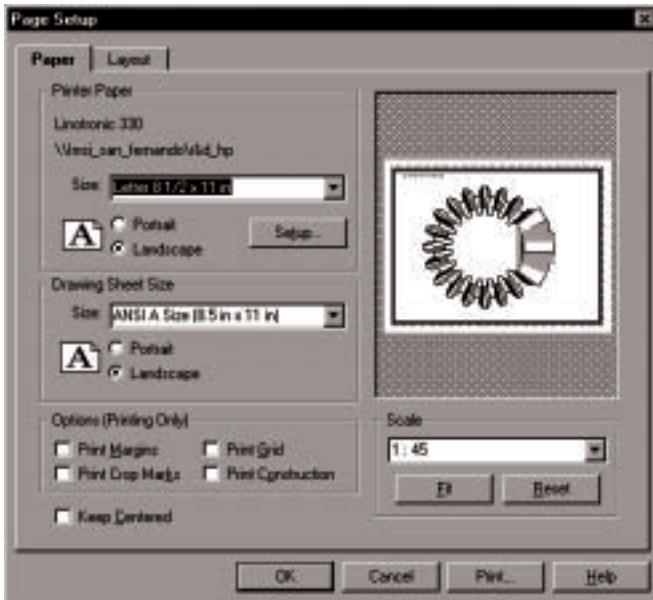
Paper property sheet of Page Setup dialog

2. In the Printer Paper control group, specify the size of the sheets of paper that you want to print on and whether you want the sheets of printer paper to be oriented in landscape or portrait.

3. In the Sheet Size control group, specify the size of the area on which you want to print the drawing and its orientation. You can do this in any units listed in the Tools|Units and Scale property sheet. For example, if you want a printout 2 feet tall and 3 feet wide, you would type 2 ft x 3 ft in the list box, and choose the Landscape option.

4. Click the Fit button to place your drawing on the tiled sheets.

5. Click the Print button to print your drawing immediately, or click OK and then choose File|Print if you want to go through the Print dialog.

NOTE: *You can also set the sheet size in either World or Page units by specifying values in the Height and Width fields on the Layout property sheet.*

Tiled Printing by Setting Rows and Columns
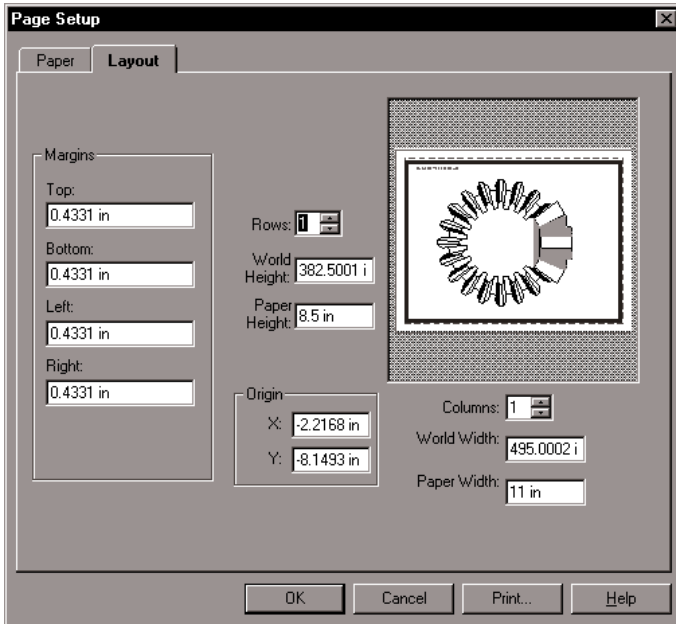
Layout property sheet of Page Setup dialog

In the Layout property sheet of the Page Setup dialog, you can specify the number of rows (the number of sheets of printer paper from top to bottom) and columns (the number of sheets of printer paper from left to right). You can then return to the Paper property sheet to fit your drawing onto the tiled printer paper. *KeyCAD* will automatically adjust the dimensions of the drawing sheet to accommodate changes in the number of rows and columns.

Command Reference

The following table summarizes the commands and dialogs accessible from the File menu:

Command	Function
New	Start a new *KeyCAD* drawing.
Open	Open a drawing in one of several file formats.
Close	Close the current drawing.
Save	Save the current drawing to disk.
Save As	Save the current drawing under a new name.
Save All	Save all open drawings.
Libraries	Bring up the Symbol Libraries property sheet of the Drawing Setup dialog.
Summary Info	View or edit document information about the current drawing.
Page Setup	Choose options for printing your drawing.
Print	Print the current drawing.
Send	Send current drawing over network using Microsoft Mail.
Recent File	Re-open one of the four most recently used files.
Exit	Exit *KeyCAD*

New

Menu: File|New   **Hotkey:** <Ctrl>+<N>   **Toolbar:** Standard

Use the New command to start a new *KeyCAD* drawing.

Create from Template Dialog

Templates are a special type of drawing file that provide a basis for a new drawing. Templates preserve all of the settings that are saved with the

drawing, such as unit and scale options, angle conventions, grid settings, and colors, as well as symbol libraries. If the Prompt for Template Name option in the General property sheet of the Program Setup dialog is checked when you choose the New command, then the Create from Template dialog will be displayed.

Create from Template dialog

When the dialog opens, it will display the templates that are stored in the current *KeyCAD* Template folder. Click on a template then click OK to use the template as a basis for your drawing. Depending on which template you choose, *KeyCAD* may run a wizard that will assist you in making additional choices.

The Create from Template dialog contains these additional controls:

- *Don't Ask Again.* Check this option if you want to bypass this dialog when you start a new drawing. The default template will then be used silently.

- *Set Default.* Click this button if you want the template you are now opening to become the default template (that is, the template that will be used if no other template is chosen). The default template is highlighted when this dialog opens.

- *Show Preview.* Check this box if you want *KeyCAD* to display a preview of the template in the Preview box. Many templates may not have a preview.

- *Description.* This area displays information about the currently selected template.

TIPS: *You can change the default template folder using the File Locations property sheet of the Program Setup dialog.*

If you create a single template that you want to use most of the time, check both the Don't Ask Again and Use as Default options, then open the template. The new template will become the default, and the dialog will no longer be displayed when you start a new drawing.

NOTE: *If you choose Don't Ask Again in the dialog box, you can reverse this setting in the General property sheet of the Program Setup dialog. Choose Tools|Program Setup and check the Prompt for Template Name check box.*

SEE ALSO: *Program Setup Dialog, Save As Dialog*

Open Dialog

Menu: File|Open   **Hotkey:** <Ctrl>+<O>   **Toolbar:** Standard

The Open command displays the Open dialog, which allows you to open an existing *KeyCAD* drawing, or a CAD drawing in any of several different standard formats. The appearance and features of the Open dialog differ significantly in Windows 3.1 and Windows 95.
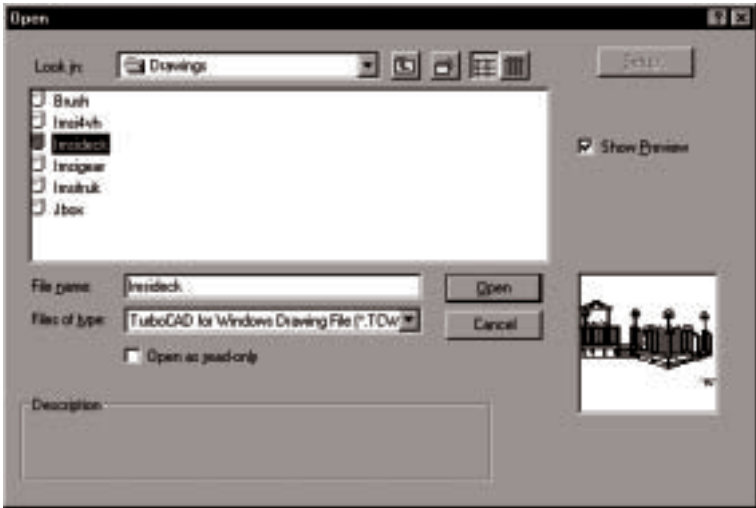
Open dialog in Windows 95

This is a variation of the Windows Open dialog, in which you use standard Windows navigation to choose a template. To open a drawing:

1. Use the standard Windows controls to navigate to the folder where the file is located.

2. Use the Files of Type control to choose the file types to display in the File Name list box. If you do nothing with this control, then only standard *KeyCAD* drawing files (.KCW) will be displayed.

3. Select the file that you want to open, then press <Enter>.

The Open dialog also provides the following controls:

Read-Only. Check this option if you wish to view a file while making sure that you don't inadvertently change it.

Description. This area displays text saved in the Subject field of the Summary Info dialog. You can place information in this field to help you identify and categorize your drawings.

Show Preview. Check this option to display a thumbnail of the drawing you have selected in the preview area of the dialog. Only *KeyCAD* (.KCW) files can have previews.

SEE ALSO: *Save As, Supported File Types*

Close

Menu: File|Close

The Close command closes the current file. If the file has been changed since it was last saved, *KeyCAD* will ask you whether you want to save the file before closing.

SEE ALSO: *Close All*

Save

Menu: File|Save   **Hotkey:** <Ctrl>+<S>   **Toolbar:** Standard

The Save command saves the current drawing to disk, using the folder and file name in which the file was previously saved.

If this is the first time you have saved the file, this command will display the Save As dialog, prompting you for a name and location for the file. If you have checked Prompt for Summary Info in the General property sheet of the Program Setup dialog, the Summary Info dialog will also appear.

TIP: *To create a new copy of the file, use File|Save As.*

SEE ALSO: *Save As, Summary Info Dialog*

Save As Dialog

Menu: File|Save As

The Save As command displays the Save As dialog, which lets you save the current drawing using a file name and folder that you specify. The appearance and function of the Save As dialog differ significantly in Windows 3.1 and Windows 95. See your Windows documentation for details.
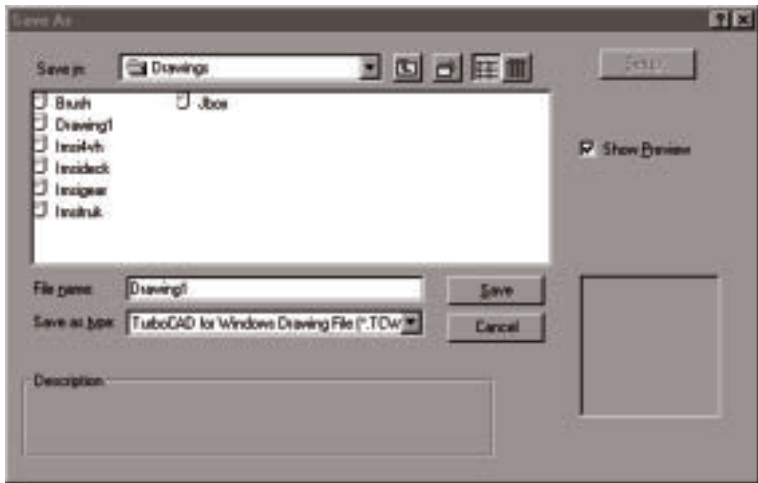
Save As dialog

To save a drawing in a specific folder and under a specific file name:

1. Use the Save In box to navigate to the folder where you want to save the file.

2. Type a name for the file in the File Name text box.

3. Use the List Files of Type control to choose the type of file you want to save. If you do nothing with this control, then the file will be saved as a normal *KeyCAD* (.KCW) file.

4. Press <Enter> to save the file.

If you have checked Prompt for Summary Info in the General property sheet of the Program Setup dialog, the Summary Info dialog will appear.

TIP: *Other graphics programs are most likely to be able to read your KeyCAD drawing if it is saved as a .DXF file. This has become the standard file exchange type for CAD drawings.*

SEE ALSO: *Save, Save All, Summary Info Dialog, Supported File Types*

Save All

Menu: File|Save All

The Save All command saves all open drawing files. If any of the files

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have not been previously saved, *KeyCAD* will prompt you for a folder
and file name for those files.

If you have checked Prompt for Summary Info in the General property
sheet of the Program Setup dialog, the Summary Info dialog will appear
once for each previously unsaved drawing.

SEE ALSO: *Save, Save As*

Libraries

Menu: File|Libraries   **Toolbar:** Standard

The Libraries command brings up the Symbol Libraries property sheet of
the Drawing Setup dialog.

SEE ALSO: *Symbol Libraries Property Sheet*

Summary Info Dialog

Menu: File|Summary Info

The Summary Info command opens the Summary Info dialog, in which
you can record information that will be saved with your drawing file.
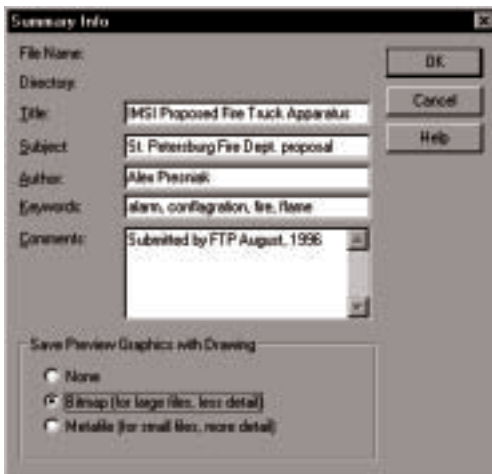
Use the text fields of the dialog box to record information about your drawing. The contents of these fields could be useful in helping you to identify drawings or versions of drawings. It can also potentially be used by Windows file search utilities.

NOTE: *If a drawing is going to be used as a symbol, then Title should be a short description of the symbol, and Subject should be a more detailed description.*

The Save Preview Graphics with Drawing options let you choose the type of graphic image that will be stored with the drawing, so that it can be previewed in *KeyCAD* and in other Windows programs:

None. Choose this option if you do not want preview graphics to be saved with your drawing.

Bitmap. Choose this option to store the preview graphic as a bitmap. This option is better suited to large files because the bitmap will use the same amount of space no matter how complex the drawing is.

Page Setup Dialog

Menu: File|Page Setup   **Toolbar**: Standard

The Page Setup dialog provides a rich set of features that let you determine exactly how drawings will be printed. It consists of two property sheets: Paper and Layout.

TIP: *After you have selected options in these property sheets, you can either make the changes in your drawing by clicking the OK button at the bottom of the dialog, or print the drawing from the dialog without making the changes in your drawing by clicking the Print button. If you click the OK button, you can then print the drawing by choosing File|Print or clicking the Print button on the Standard toolbar.*

Paper Property Sheet

The Paper property sheet lets you determine the size and orientation of the printer paper and the drawing sheet.

Page Setup — Paper / Layout tabs

Printer Paper

Linotronic 330

\\lmsi_san_fernando\r&d_hp

Size: Letter 8 1/2 x 11 in

A ○ Portrait   Setup...
   ● Landscape

Drawing Sheet Size

Size: ANSI A Size (8.5 in x 11 in)

A ○ Portrait
   ● Landscape

Options (Printing Only)

☐ Print Margins   ☐ Print Grid
☐ Print Crop Marks ☐ Print Construction

☐ Keep Centered

Scale

1 : 45

Fit   Reset

OK   Cancel   Print...   Help
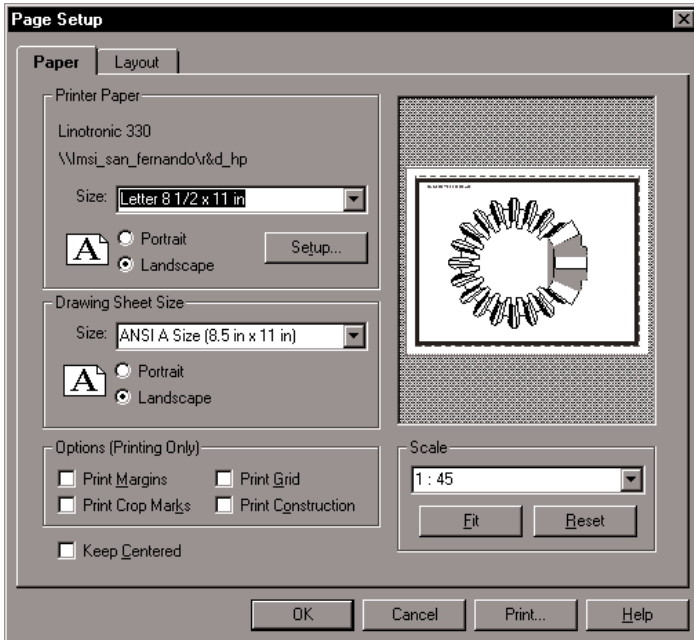
Paper property sheet

The printer paper is the physical paper that goes through the printer; the drawing sheet is the area on which your drawing will be printed, which can span multiple sheets of printer paper. This property sheet also contains a variety of option controls and a function for viewing or setting the scale of the drawing.

SEE ALSO: *Print Setup Dialog, Printer Paper and the Drawing Sheet, Layout Property Sheet*

Printer Paper

The Printer Paper control group lets you set the size and orientation of the printer paper. These are the same settings that you can control through the Print Setup dialog, so changes that you make in this dialog will be reflected in Print Setup, and vice versa. To set the paper size and orientation:

1. Click in the Size list box and choose a standard paper size.

2. Choose either portrait or landscape orientation by clicking one of the option buttons. The icon to the left of the option buttons will change to reflect your choice.

Clicking the Setup button in this control group brings up the Print Setup dialog, in which you can choose another printer or access functions that directly control your printer. This dialog is specific to your printer driver, so check your printer or printer driver documentation for details.

Drawing Sheet Size

The Drawing Sheet Size control group lets you control the size and orientation of the area on which your drawing will be printed. This area can be imposed on a single sheet of printer paper, or it can span multiple sheets of printer paper. Its orientation can also be set separately from the orientation of the printer paper. For a description of how printer paper and drawing sheet size are related, see Printer Paper and the Drawing Sheet. To set the size and orientation of the drawing sheet:

1. Click in the Size list box and choose a standard paper size, or type in a custom size using standard measurements (for example, **2 ft x 2 ft**).

2. Choose either portrait or landscape orientation by clicking one of the option buttons. The icon to the left of the option buttons will change to reflect your choice.

The page display on the right will change to reflect the relationship between printer paper and drawing sheet size and orientation.

Options (Printing Only)

The buttons in the Options (Printing Only) control group let you determine what elements are printed. These options affect only the printing of the drawing and are not reflected in the appearance of the drawing on screen.

Print margins. Check this button to print a border on the margin of the printer sheets. This is useful for framing the drawing if it is printed on a single sheet.

Print crop marks. Check this option button to print crop marks on the printer sheets. This is useful if you want to print across multiple printer sheets and crop the sheets so that they can be pasted against a backing.

Print grid. Check this option if you want to print grid marks. The grid will be printed if it is currently displayed.

Print construction. Check this option if you want to print constructions lines and circles.

Page Layout Scale

The controls in the Scale control group let you scale the drawing so that it fits within the drawing sheet or choose a custom scale.

To scale the drawing so that it fits into the drawing sheet, click the Fit button. The drawing will be centered as it is fit onto the drawing sheet.

To choose a custom scale, click in the list box and choose a pre-set scale, or type a scale directly into the list box in the form **1 in = 1 ft**. When typing custom scales, you can use any measurement available in the Units and Scale property sheet of the Drawing Setup dialog (available from the Tools menu). You can click the Reset button at any time to return the scale that was set when you entered the Page Layout dialog.

Keep Centered

Check the Keep Centered option to keep the drawing centered over the drawing sheet.

Layout Property Sheet

The Layout property sheet gives you direct control of the number of rows (sheets of printer paper arranged vertically) and columns (sheets of printer paper arranged horizontally). It also has functions for setting the printer paper margins, setting the height and width of the drawing sheet in either World or Paper units, and relocating the paper in the drawing area.
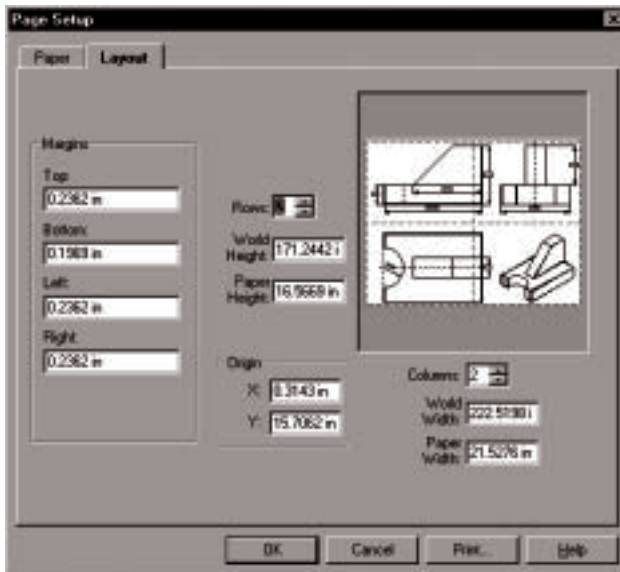
Layout property sheet of Page Setup dialog

Margins

Margins, the dotted rectangle inside the edges of the paper, let you know whether you are drawing inside or outside the printable area of your drawing.

The margins control group contains four text boxes in which you can specify the top, bottom, left, and right margins of the printer paper in Paper units. By default, the margins are displayed in units set in the Units and Scale property sheet, but you can enter values in other units as well.

NOTE: *You cannot set the margins of the drawing sheet independently of the margins of the printer paper.*

TIP: *If you set all the margins to zero, KeyCAD will offer to set them back to give you the biggest possible printable area on the current printer.*

Rows and Columns

The Rows and Columns controls let you set the number of rows and columns of printer sheet paper, for tiled printing. As you increase the number of rows and columns, the size of the drawing sheet—the area on which your drawing will be printed—increases correspondingly. The Paper Height and Paper Width controls let you directly set the height and width of the drawing sheet. As you increase these values, *KeyCAD* will automatically increase the number of rows and columns to accomodate the change in the drawing sheet size.

Origin

The Origin boxes let you move the paper to a different location in the drawing. You do this by specifying the location of the lower-left corner of the paper in absolute coordinates: Type the horizontal coordinate into the X box and the vertical coordinate into the Y box.

TIP: *This feature is useful if you want to print a particular area of a large drawing by relocating the paper to the area that you want to print.*

SEE ALSO: *Coordinate Systems, Units and Scale Property Sheet*

Print Dialog

Menu: File|Print   **Hotkey**: <Ctrl>+<P>   **Toolbar**: Standard

The Print dialog provides a set of functions which include printing a
single view of a drawing, or printing specific sheets in a drawing that
spans multiple sheets of printer paper. The controls on the dialog differ
slightly in Windows 95 and Windows 3.1.
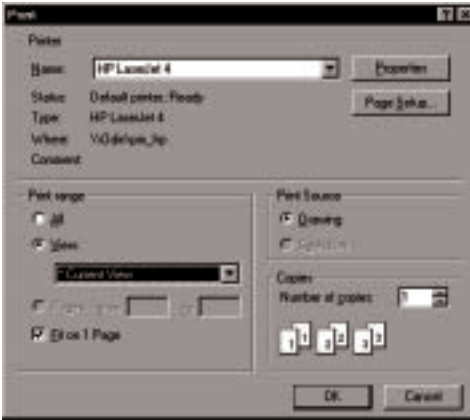
Print dialog in Windows 95

Print dialog in Windows 3.1

SEE ALSO: *Page Setup Dialog, Printing, Views of the Drawing Space*

Printer (Windows 95)

The Printer control group lets you select any printer that is available on your system and access its properties. Click in the Name dropdown box to select an available printer. Click on the Properties button to access the current printer's property sheets. Click on the Page Setup button to go to the Page Setup dialog.

Print Source

The Print Source control group lets you choose whether to print all of the drawing objects, or only those that are currently selected.

NOTE: *You can control the printing of other elements, such as crop marks, margins, and constructions, in the Paper property sheet of the Page Setup dialog.*

Print Range

The Print Range control group provides options for setting a specific view of the drawing to print, and for selecting a range of pages when printing a drawing that spans multiple printed sheets.

TIP: *This is useful for printing tiled drawings if you only want to re-print specific pages.*

All. Select this option if you want to print all of the current drawing. If you have chosen Selection in the Print Source control group, only objects that are selected will be printed. (If your drawing spans multiple printer sheets, the total number of sheets will be shown in this option label.)

View. Select this option to print a single view of the drawing. You can print the current view by default, or click in the list box and choose any named view.

Pages. Select this option to print a range of pages if the Page Layout dialog is set up to print multiple pages. Specify the first page to be printed in the left text box, and the last page to be printed in the right text box. *KeyCAD* counts the printer sheets starting in the lower-left corner from left to right, and from bottom to top.
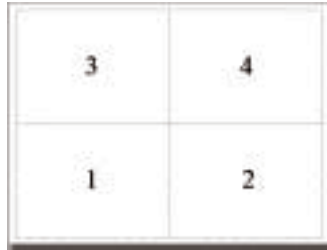
Count pages from left to right, and from bottom to top

Fit on 1 page. Check this box if you want to print everything on a single page. This option is compatible either with printing a view, or with printing the entire drawing. You can't have this option checked and the Range option turned on at the same time.

Print Quality (Windows 3.1 only)

The contents of this list box will vary, depending on settings available in your current printer. In some cases the list box may contain specific settings, such as 300 or 600 DPI (Dots Per Inch). In other cases, the list box may contain High, Medium, Low, and Draft settings. Higher quality print settings produce finer resolution; lower quality print settings print out more quickly.

Copies

Type the number of copies of the drawing that you want to print in the copies text box.

Collate Copies (Windows 3.1 only)

Check this box to instruct the printer to collate multiple copies of a drawing that consists of more than one page.

Print Setup Dialog (Windows 3.1 only)

The Print Setup button brings up the Print Setup dialog.

The Print Setup dialog enables you to choose a printer, if there are

4

Editing Objects

Edit Menu and Tools

The Edit menu contains commands and tools for selecting, editing, and copying objects. These features enable you to:

- Undo a previous operation, or redo what was previously undone.

- Perform standard Windows clipboard operations: Cut, Copy, and Paste.

- Clear (delete) objects using several options.

- Select objects with the Select tool, or categorically by type, layer, or several other criteria.

- Create multiple copies of entities along lines, around arcs, and in two-dimensional arrays.

- Edit OLE objects using the source application and make changes to how OLE objects are linked to your *KeyCAD* drawing.

In addition to the features available from the Edit menu and toolbars, *KeyCAD* has two separate modes for mouse-based editing:

- Select Edit, in which you scale, rotate, or move objects.

- Node Edit, in which you reshape entities by editing special points called *nodes*.

The first part of this chapter will focus on mouse-based editing. The remainder of the chapter provides a complete reference to the commands and tools available from the Edit menu.

Editing with the Mouse and Local Menu

KeyCAD offers powerful capabilities for editing directly with the mouse and local menu (which you access by right-clicking in the drawing area). There are two distinct editing modes that you can use for this purpose: Select Edit and Node Edit.

NOTE: *Select Edit mode is essentially similar to Handle-based editing in KeyCAD 2.0.*

Select Edit

Select Edit is the editing mode an object is in when you first select it. In this mode, the selected objects are bounded by a *selection rectangle* that has handles on each corner and at the midpoint of each side. At the center of the selection rectangle is the *reference point*, which is connected to the *rotation handle,* located just outside of the selection rectangle.
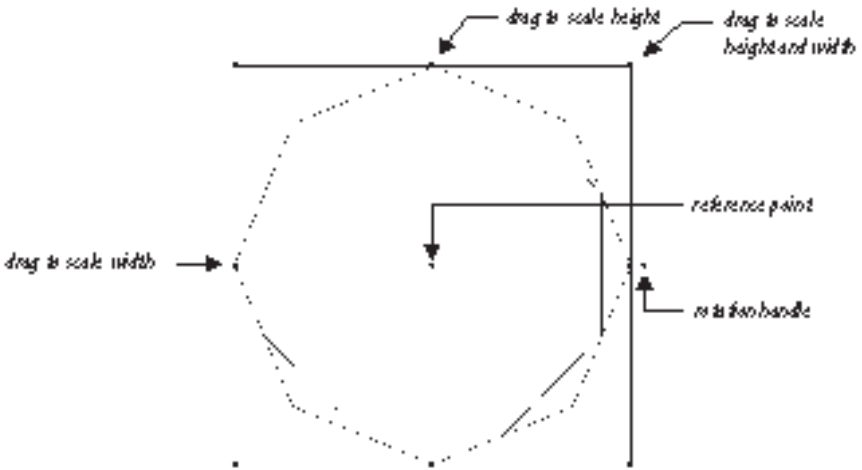
Selected objects in Select Edit mode

SEE ALSO: *Geometric and Cosmetic Select Modes*

Tip: You can get information about selected objects by choosing View|Selection Info.

Geometric and Cosmetic Select Modes

KeyCAD offers two distinct selection modes: Geometric Select mode and Cosmetic Select mode. You can switch between the two modes by clicking the Geometric Select Mode check box, located in the General property sheet of the Tools|Program Setup dialog.

In order to understand the difference between these selection modes, you must first realize that every *KeyCAD* entity has two separate sets of extents: *geometric extents* and *cosmetic extents*. Geometric extents are calculated based on the distance between the specific vertices that define the entity geometrically (for example, the three corners of a triangle, or the two endpoints of a line segment). Cosmetic extents are calculated based on distances between the outermost points on the object that will be drawn on the screen or on the printer. In many cases, the geometric and cosmetic extents will be exactly the same. This is true, for example, if you measure the extents of a single line segment drawn with the minimum (default) pen width. If, however, you are working with a double-line entity, or with an entity that is drawn with a pen that has greater than the default width, the geometric and cosmetic extents will differ.

Advantages of Geometric Select Mode

Other drawing packages use cosmetic selection only. What is the advantage of geometrically accurate selection? There are two main advantages:

- When you scale using geometric extents, it is easy to see exactly what is going to be scaled. If, for example, you scale a double-line entity, *KeyCAD does not* scale the width of the double line, just as it does not re-scale the width of the pen. Geometric Select mode makes it visually obvious that this is the case.

- When you scale an orthogonally oriented linear entity using geometric extents, you get accurate results. If, for example, you scale a 1-inch horizontal line segment 100 percent along the X axis, the line segment will always be 2 inches long. If you were to use cosmetic extents, the results would vary, depending on the width of the pen.

The disadvantage of geometric selection is that if you snap the endpoint of a wide line to another object, the cosmetic end of the line will overlap the object. You can work around this problem by selecting a line using cosmetic extents and using a snap mode to position the cosmetic end of

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the line.

TIPS: *If you want to scale the width of a double line as you scale an entity, you need to first turn the double line into two single lines. To do this, select the entity, then choose Explode from the Format menu. If the double line is contained within a group, you will have to choose Explode more than once, because the Explode command breaks an object down into its constituent parts one step at a time.*

Another way to adjust the width of a double line, or to change the width of the pen, is to select the entity and double-click on it, bringing up its properties sheets. You can change the width of a double line in the Double Line property sheet, or the width of the pen in the Pen property sheet.

SEE ALSO: *Explode, Properties Dialog*

Moving the Selection's Reference Point

You can move the reference point of a selection to any position in the drawing area. To do this, hold down the <Ctrl> key while clicking on the reference point. The cursor will change to a hand grasping the point. You can now move the reference point to any position by defining a point. You can do this with the mouse using any snap mode, or you can type the coordinates of the point into the Coordinate Fields and press <Enter>.

Ctrl-Click on reference point   repsosition reference point using any snap mode
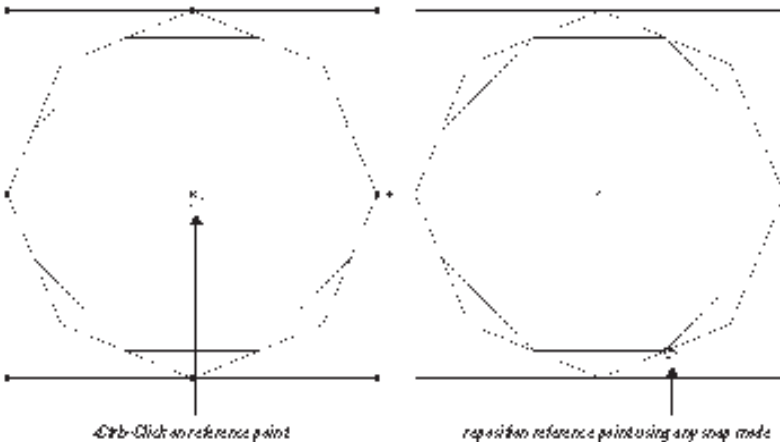
TIPS: *When you rotate an object, you do so in relation to its reference point. This means that you can rotate the object not only around its own center, but around any arbitrary point or object in the drawing space. For example, you can use Snap to Arc Center mode to move the reference point of the selection to the geometric center of a circle or arc, then "orbit" the selection around that point.*

To restore the reference point to its original location at the center of the selection, you can deselect and then reselect the objects.

Moving a Selection

There are three ways to move a selection in Select Edit mode: "picking it up" by its reference point, dragging it by its reference point (OLE drag and drop), and "simple dragging" (that is, dragging by any point within the selection rectangle other than the reference point).

"Picking" the Reference Point

In most cases the most convenient and accurate way to move an object is to click on its reference point, then define a new location for the reference point. The object moves with the reference point:

1. Select the object or objects that you want to move.

2. If desired, move the selection's reference point relative to the selection. For example, if in moving a rectangle you wish to precisely place its lower-left corner, rather than its center, first move the reference point to the lower-left corner of the rectangle. You can use snap modes for precise placement of the reference point.

3. Click on the selection's reference point. The cursor changes to four-directional pointer (crossed arrows).

4. Define the point where you want to place the selection. You can do this by clicking the mouse, using any snap mode. Alternatively you can press <Ctrl>+ <R> to activate the Coordinate Fields, then type the coordinates of the desired location in the Coordinate Fields and press <Enter>.

TIP: *When you move a selection by picking its reference point, you can use Ortho mode when you place the selection. You can turn on Ortho mode in the menu, toolbar, or local menu before or during the placement of the selection; you can also toggle Ortho mode during placement by holding*

down the <Shift> key.

SEE ALSO: *Moving the Selection's Reference Point*

OLE Drag and Drop

You are using OLE drag and drop whenever you drag an object by its reference point.

1. If you're going to use a snap mode, set it before starting the move. You may also want to move the reference point relative to the selection before beginning the move.

2. Place the mouse cursor directly over the reference point. You will know it is there when the mouse cursor turns into a four-directional pointer.

3. Hold the mouse button down while "dragging" the object to the new location, then release the mouse.

The main advantage of OLE drag and drop is that it lets you *copy* the selection from one drawing into another, or even from a *KeyCAD* drawing into another application.

WARNING: *The behavior of snap modes is slightly unusual when you are moving an object using OLE drag and drop. If KeyCAD finds a snap point based on the location of the mouse at the point of release, then the reference point will snap to the snap point. If KeyCAD cannot find a snap point, then the object will simply be dropped at the point of release.*

SEE ALSO: *Snap Modes*

Simple Dragging

If you place the mouse pointer anywhere inside a selection other than on the reference point, you can simply drag the selection to another location within the drawing. Simple dragging does not allow you to use snap modes or the Coordinate Fields for accurate placement, but it is convenient if you don't need accurate placement.

Scaling a Selection

You can scale a selection either by dragging its handles, or by entering values for its X and Y scale in the Edit Bar.

Drag the handles on the corners to freely resize the selection both horizontally and vertically. To scale the selection only horizontally or only vertically, drag the handles on the sides, top, or bottom of the

selection rectangle. To scale the selection while keeping the same proportions of height and width, hold down the <Shift> key while dragging the handles.

To scale a selection using the Edit Bar, type values into the XScale and YScale Edit Bar fields, then press <Enter>. For example, if you want to double the width of the selection while increasing its height by 50 percent, type 2 into the XScale field and type 1.5 into the YScale field, and then press <Enter>.

NOTE: *If the object you are scaling contains double lines or lines drawn with a pen width greater than 0, the results of scaling a selection will vary, depending on whether you are using geometric or cosmetic selection. For geometrically accurate results, use the default Geometric Select mode.*

SEE ALSO: *Geometric and Cosmetic Select Modes*

Copying While Moving or Scaling

The Make a Copy option in the local menu lets you make a copy of the selection at the same time that you move or resize the object. To use this option:

1. Select the object or objects that you want to copy.

2. Right-click in the drawing area to open the local menu, then choose Make a Copy, if it isn't already checked. (This is a toggle, so if it already has a check mark next to it, choosing it will turn it off.)

3. Move or scale the selection by either picking the reference point or dragging by the reference point.

When you move or scale the selection, the original will remain at its original location and size, while a copy is created at the new location and size.

TIP: *You can turn on the Make a Copy option temporarily by holding down the <Ctrl> key after you start to copy or scale the selection.*
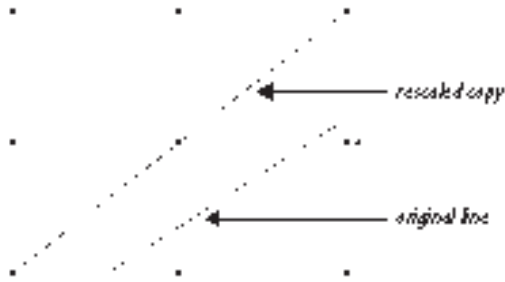
Copying a selection with Make a Copy

Rubber Stamp

Turning on the Rubber Stamp option allows you to place multiple copies of selected objects in your drawing. To use this option:

1. Select the object or objects that you want to copy.

2. Right-click in the drawing area to open the local menu, then choose Rubber Stamp. The cursor will change into a rubber stamp symbol.

3. Click anywhere in the drawing to place a copy of the selection.

You can continue clicking to place as many copies of the selection as you like. To turn Rubber Stamp off, right-click in the drawing area to open the local menu and choose Cancel, or press <Esc>.

NOTE: *Rubber Stamp placement does not respond to snap modes.*

Rotating a Selection

To rotate a selection either move its rotation handle, or type a value into the Edit Bar.

To move the rotation handle of the selection, click on the handle, then define a point by clicking the mouse, using any snap mode. To rotate the selection using the Edit Bar, type a value for the degrees of rotation into the Edit Bar's Rotation field, then press <Enter>. The value you enter in the Edit Bar is the number of degrees that the selection will be rotated in the positive direction of rotation. To rotate the object in the opposite direction, enter a negative value in the Rotation field.

NOTE: *The positive direction of rotation is counterclockwise by default. You can reverse the direction in the Angle property sheet of the Drawing Setup dialog.*

TIP: *When you rotate a selection you can use Ortho mode to constrain the angle of rotation. You can turn on Ortho mode in the menu, toolbar, or local*

menu before or during the placement of the selection; you can also toggle Ortho mode during placement by holding down the <Shift> key.

rotation handle ————
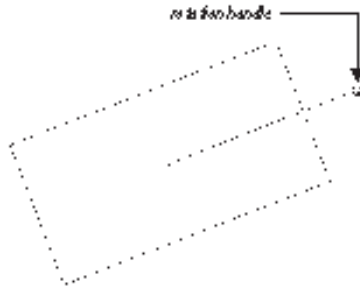
Rotating a selection

SEE ALSO: *Ortho*

Node Edit

In Node Edit mode you can reshape entities by directly manipulating particular points called *nodes.*

- Add, delete, and move nodes on entities composed of line segments (including lines, double lines, polygons, and sketches).

- Divide line segments into any number of subsegments of equal length.

- Break polygons, making them into multilines.

- Close multilines, making them into polygons.

- Close the endpoints of double lines.

- Reshape Bezier and spline curves by adding, deleting, and moving nodes on the line segments that form the curves' frame.

- Change the radius of circles and circular arcs.

- Change the major and minor axes of ellipses and elliptical arcs.

- Change the start and end angles of circles, ellipses, and arcs.

- Close arcs, making them into circles.

- Divide circles, arcs, and ellipses into arc segments of equal angular length.

To activate Node Edit mode, first select the entity that you wish to edit, then click the right mouse button in the drawing area to display the local menu. From the local menu, choose Node Edit. The appearance of the

selection will change. Instead of a selection box with handles, you will see highlighted objects with nodes displayed as blue rectangles. The number and placement of the nodes depends on the type of object selected.

To deselect an entity in Node Edit mode, click anywhere in the drawing area other than on the selected object.

NOTE: *You can only edit one entity at a time in Node edit mode. You cannot use this mode to work with multiple selected entities or with groups. To edit the nodes of an entity that is in a group, first select the group, then break it into its component entities using the Format/Explode command. Because groups can be nested, you may need to explode the selection more than once in order to isolate a single entity.*
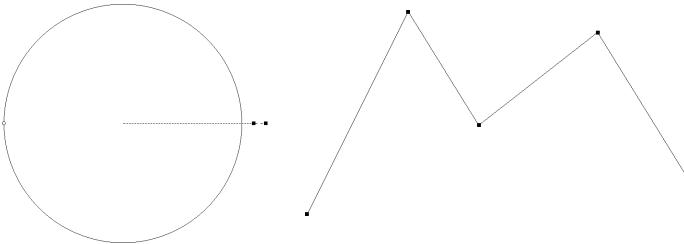
Selected objects in Node Edit mode

Node Edit with Linear Entities

There are several editing actions you can perform on entities composed of line segments. These actions include adding a node, deleting a node, moving a node, and dividing a line or double-line segment.

Adding a Node to a Line or Double Line Segment

To add a node to a single or double line segment on a line, multiline, or polygon:

1. Select the entity that you want to edit. Click the right mouse button in the drawing area to open the local menu, then choose Node Edit.

2. Hold down the <Ctrl> key while placing the mouse cursor over a part of a line or double-line segment where there is no node. When you are in the correct position, the cursor will show a node being placed at the center of a cross.

3. Click to place the node on the line segment.

A node will appear on the line or double-line segment at the location of the mouse cursor. You can now drag the node to reshape the line segment into a multiline.

NOTE: *As an alternative to holding down the <Ctrl> key while clicking on the line segment, you can place the mouse cursor at the position where you want to add the node, then right-click and choose Add Node from the local menu.*
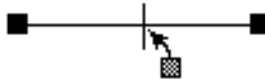
Adding a node to a line segment

Deleting a Node from a Line Segment

To delete a node from a line or double line segment on a line, multiline, or polygon:

1. Select the entity that you want to edit. Click the right mouse button in drawing area to open the local menu, then choose Node Edit.

2. Hold down the <Ctrl> key while placing the mouse cursor over the node that you want to delete. When you are in the correct position, the cursor will change to an icon showing a node being thrown into a trash can.

3. Click to delete the node.

If you remove a node from a polygon, *KeyCAD* will keep the polygon closed by connecting the opposite endpoints of the line segments that were connected at the deleted node.

NOTE: *You cannot delete a node that is necessary to define an entity as a line segment or polygon. This means that you cannot delete the endpoint of a single line segment, and you cannot delete any node of a triangle. If you delete the endpoint of a multiline, you will delete the entire line segment that terminates at that point.*
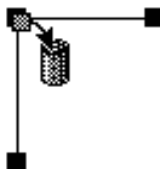
Deleting a node

SEE ALSO: *Closing Multilines, Breaking Polygons*

Moving a Node

You can reshape a single line or double line segment on a line, multiline, or polygon, by defining a point for the location of any node. To do this:

1. Select the entity that you want to edit.

2. Click the right mouse button to open the local menu and choose Node Edit.

3. Click on the node that you want to move.

4. Define the point to which you want to move the node. You can do this either using the mouse and any snap mode or by typing the coordinates of the point in the Coordinate Fields and pressing <Enter>.

NOTE: *You can also simply drag the node to any position you want.*

Dividing a Line or Double Line Segment

The Segment Divide command from the local menu lets you add nodes to a line segment, breaking the line segment into a multiline composed of segments of equal length. To do this:

1. Select the entity that you want to edit.

2. Click the right mouse button to open the local menu and choose Node Edit.

3. Place the mouse cursor over the line segment that you want to divide.

4. Click the right mouse button to open the local menu and choose Segment Divide. *KeyCAD* displays the Divide Segment By dialog.

5. Enter the number of parts into which you want to divide the line into the dialog, and then press <Enter>.

Breaking Polygons

Node Edit allows you to break a polygon at any node, turning it into a multiline. To break a polygon:

1. Select the polygon that you want to edit.

2. Click the right mouse button to open the local menu and choose Node Edit.

3. Position the cursor over the node where you want to break the polygon.

4. Click the right mouse button to open the local menu and choose Break.

The polygon is now an open multiline, and there are two nodes located at the point where you broke the polygon. You can open the polygon visually by moving these nodes apart.

Note: You can also use this technique to break a closed Bezier or Spline curve.

Closing Multilines

Node Edit allows you to close a multiline, making it into a polygon. To close a multiline:

1. Select the multiline entity that you want to close.

2. Click the right mouse button to open the local menu and choose Node Edit.

3. Position the cursor over either endpoint of the multiline.

4. Click the right mouse button to open the local menu and choose Close.

KeyCAD will draw a line between the two endpoints of the multiline (assuming that these two nodes don't share the same location), forming a geometrically closed polygon. If the two endpoints of the multiline share the same location, *KeyCAD* will simply close the polygon geometrically, since it is unnecessary to draw a line between the two points.

Note: You can also close a Bezier or Spline curve using this technique.

WARNING: *If you simply snap the endpoints of a multiline together you have something that looks exactly like a polygon; however, KeyCAD still considers it a multiline. This means that you won't be able to use a fill by setting the entity's brush color and pattern in its Properties sheet. In order to turn a multiline into a true polygon, you must close the multiline.*

Closing Endpoints of Double Lines

Node Edit allows you to close the endpoints of double lines. To do this:

1. Select the double-line entity.

2. Click the right mouse button to open the local menu and choose Node Edit.

3. Position the cursor over either endpoint of the double line.

4. Click the right mouse button to open the local menu and choose Close Double End.

KeyCAD will close the double line with a flat end.

Editing Double Line Cutlines and Gaps

Cutlines and gaps occur at the intersections of double lines. For example, if you form a clean T intersection of two double lines at an angle other than 90 degrees, the end of one of the double lines will be "cut" at an angle; the other double line will have a "gap" to accommodate the first double line. Node Edit mode allows you to drop the cutlines in the first line, "evening" out the end of the line; it also allow you to fill in the gap in the second line.
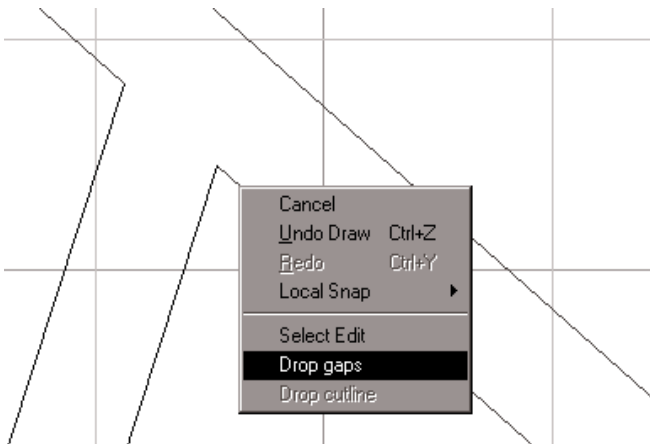
Closing a gap formed at a T intersection

Node Edit with Curves and Sketched Lines

Sketches (entities created with the Sketch tool) are actually multilines composed of minuscule line segments, so they can be edited in Node Edit mode as multilines that have a very large number of nodes.

Bezier and spline curves are both drawn in reference to a sequence of points (spline curves actually pass through the points, while Bezier

curves gravitate toward the points). In Node Edit, each of the defining points of the curve is represented by a node.

In Node Edit mode, you can move or delete any of the nodes that define a curve, and you can add new nodes that you can move to reshape the curve. You can move and delete the nodes on a curve the same way that you move and delete nodes on a line segment, multiline, or polygon.

You can also close a Bezier or Spline curve, or break a closed Bezier or Spline curve at any node, using the same technique that you would use to close a multiline or break a polygon.

These techniques will be discussed in the following two sections.

SEE ALSO: *Adding a Node to a Line or Double-Line Segment, Breaking Polygons, Closing Multilines, Deleting a Node, Moving a Node*

Adding a Node to a Curve

To add a node to a Bezier or spline curve:

1. Select the curve that you want to edit. Click the right mouse button in drawing area to open the local menu, and choose Node Edit.

2. Hold down the <Ctrl> key while placing the mouse cursor over a part of the curve where there is no node. When you are in the correct position, the cursor will change to an icon showing a node being placed at the center of a cross.

3. Click to place the node on the line segment.

A node will appear on the line segment at the location of the mouse cursor. You can now drag the node to reshape the curve.

NOTE: *As an alternative to holding down the <Ctrl> key while clicking on the line segment, you can place the mouse cursor at the position where you want to add the node, then right-click and choose Add Node from the local menu.*

Dividing a Segment of a Curve

The Divide Line command from the local menu lets you add nodes to a line segment on the frame of a curve, breaking the line segment into any number of subsegments of equal length. To do this:

1. Select the curve that you want to edit. Click the right mouse button to open the local menu, then choose Node Edit.

2. Place the mouse cursor over the part of the curve that you want to divide.

3. Click the right mouse button to open the local menu and choose Divide. *KeyCAD* will display the Divide Segment By dialog.

4. Enter the number of parts into which you want to divide the line segment into the dialog, and then press <Enter>.

You can move any of the nodes on the curve's frame to reshape the curve.

Node Edit with Circles, Ellipses, and Arcs

In Node Edit mode you can change the radius of a circle or circular arc, the major and minor axes of an ellipse or elliptical arc, and the starting or ending angles of any arc, circle, or ellipse. You can also divide any circular or elliptical entity into any number of equal arcs.

To edit a circle, ellipse, or arc in Node Edit mode, first select the entity that you want to edit, then click the right mouse button in the drawing area to open the local menu and choose Node Edit. The entity that you selected will be highlighted with its nodes displayed. In the case of a circle or circular arc, you will see three nodes: two for manipulating its starting and ending angles, and one for manipulating its radius. In the case of an ellipse, you will see four nodes: two for manipulating its starting and ending angles, and two for manipulating its major and minor axes.

TIP: *If you are confused by the notion of editing the starting and ending angles of a circle or ellipse, try thinking of circles and ellipses as arcs that have their starting and ending angles at zero and 360 degrees.*
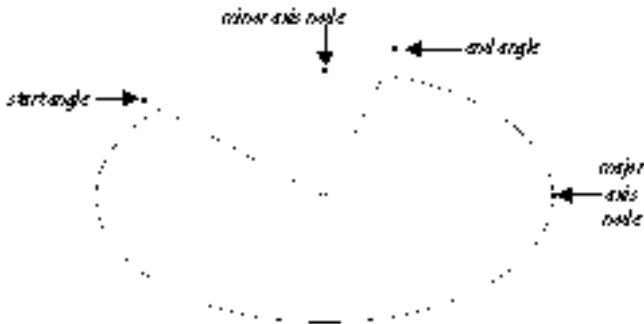
Elliptical arc in Node Edit mode

Changing the Radius of a Circle or Circular Arc

To alter the radius of a circle or circular arc:

1. Select the circular entity that you want to edit. Click the right
   mouse button to open the local menu, then choose Node Edit.
   You will see three nodes, one of which is a blue square: You can
   manipulate this *radius node* to change the circle's radius.

2. You can now either define a point for the radius node, or directly
   enter the radius of the circle in the Edit Bar. To define a point,
   click on the radius node, then define a point by clicking the mouse,
   using any snap mode; or type values for the point's coordinates in
   the Coordinate Fields, then press <Enter>.

   To specify the radius in the Edit Bar, press <Tab> to leave the
   drawing area, type a value for the radius into the Radius field of
   the Edit Bar, then press <Enter>.

You can also simply drag the radius point to another position, but in
doing this you lose the precision of using snap modes, the Coordinate
Fields, or the Edit Bar.

Editing the Axes of Ellipses and Elliptical Arcs

To scale an ellipse or elliptical arc:

1. Select the elliptical entity that you want to edit. Click the right
   mouse button to open the local menu and choose Node Edit. You
   will see four nodes, two of which are blue rectangles on the
   perimeter of the ellipse: You can manipulate these nodes to scale
   the ellipse.

2. You can now either define points for the two nodes or directly enter
   values for the lengths of the major and minor axes of the ellipse in
   the Edit Bar:

   • To define points for the nodes, first click on one of the nodes,
     then define a point either by clicking the mouse using any snap
     mode, or by typing values for the point's coordinates in the
     Coordinate Fields and pressing <Enter>. Repeat this procedure
     for the other node.

   • To specify the lengths of the axes in the Edit Bar, press <Tab> to
     leave the drawing area, type a value for the axes in the Major
     and Minor fields of the Edit Bar, then press <Enter>.

You can also simply drag the nodes to another position, using any snap
mode.

TIP: *While changing the length of an ellipse's axis, you can hold down the*

<Shift> key, constraining the proportions of the ellipse and changing the length of both axes simultaneously.

Changing the Starting and Ending Angle of an Arc

To alter the starting and ending angles of a circle, circular arc, ellipse, or elliptical arc:

1. Select the entity that you want to edit. Click the right mouse button to open the local menu and choose Node Edit. You will see three nodes for circular entities and four nodes for elliptical entities. Two of the nodes are represented as a yellow and a blue circle at the ends of dotted lines. You can manipulate these nodes to change the starting and ending angles of the entity.

2. You can now either move the nodes to specify starting and ending angles or enter values for the starting and ending angles in the Edit Bar:

    To move the nodes, first click on one of the nodes, then define a point by clicking the mouse, using any snap mode. Repeat this procedure for the other node.

    To specify the starting and ending angles in the Edit Bar, press <Tab> to leave the drawing area, type a value for the axes in the Start Ang and End Ang fields of the Edit Bar, then press <Enter>.

Dividing an Arc

The Divide command from the local menu lets you divide an arc or circle into subarcs of equal angular length. To do this:

1. Select the entity that you want to edit. Click the right mouse button to open the local menu and choose Node Edit.

2. Click the right mouse button to open the local menu, and choose Arc Divide. *KeyCAD* will display the Divide Arc By dialog.

3. Enter the number of parts into which you want to divide the arc into the dialog, then press <Enter>.

Node Edit of Dimensions

Dimensions have nodes that let you manipulate the position of the dimension line, extension lines, and dimension text. You can also move the *associative nodes* of dimensions. These are the nodes that are associated with the dimension text, so that changing the position of the nodes will cause automatic dimension text to be updated.

NOTE: *If a dimension is associated with an entity, then moving the node attached to the entity will break the association between the entity and the dimension.*
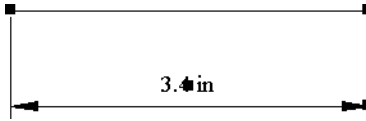
Nodes of a horizontal dimension

Manipulating Text

KeyCAD has a rich set of features for manipulating text, based on the principle that you can edit the text either as:

- a TrueType font
- a group of polygons
- individual polygons (each character becomes one polygon)

When you first create text, it is a TrueType font. You can select the text, stretch it, and rotate it. When text is in this state you can edit it by changing the Attribute field of the General property sheet of the Properties dialog. Double-click on the text with the Select tool to bring up this dialog. You can reformat the text (changing its font, size, attributes, and justification) using the Format Text toolbar or the Text property sheet of the Properties dialog.

If you want more freedom to manipulate the text graphically, you need to turn the text into an irregular polygon. To do this, select the text object using the Select tool, then choose Format|Explode. The first time you do this, you will turn the text object into a group of polygons. You can now stretch, rotate, or flip the graphic text using handle-based editing, producing a variety of effects.

Effects of manipulating text as group of polygons

If you explode the text a second time, it will ungroup so that each individual character can be manipulated as a polygon. You can resize, stretch, move, flip, or rotate each individual character in Select Edit mode. You can change the pen, brush pattern, or other properties of the multiline by double-clicking on the selection and changing the settings in the Properties dialog.

If you select an individual polygon (character), and then right-click and choose Node Edit from the local menu, you can edit individual nodes. You can move, delete, and add nodes; and you can also break the polygon into a multiline.

Polygon character in Node Edit mode

If you explode an individual polygon character, it will be broken into individual line segments.

SEE ALSO: *Node Edit of Lines..., Select Edit, Text Property Sheet*

Command Reference

The following table summarizes the commands accessible from the Edit menu:

Command	Function(s)
Undo	Reverse previous action.
Redo	Reverses the effect of the previous Undo command.
Cut	Remove selected objects and place them on the Windows clipboard.
Copy	Copy selected objects to clipboard.
Paste	Paste objects into drawing from clipboard.
Paste Special	Paste objects from clipboard as data or as OLE objects.
Clear	Remove selection, all objects, a single construction, or all constructions from drawing.
Select	Activate the Select tool.
Select All	Select all objects in active drawing.
Select By	Select by entity type, layer, or attribute.
Copy Entities	Create multiple copies of objects along lines, around arcs, in two-dimensional arrays, and more.
Edit Links	Make changes to linked OLE objects.
Edit Object	Edit OLE object using its source application.

Undo

Menu: Edit|Undo   **Hotkey:** <Ctrl>+<Z>   **Toolbar:** Standard

The Undo command reverses the previous action, restoring your drawing to its state prior to your last editing or drawing command. The name of this command will change, depending on the specific action that it undoes.

SEE ALSO: *Redo*

Redo

Menu: Edit|Redo   **Hotkey:** <Ctrl>+<A>   **Toolbar:** Standard

The Redo command reverses the effects of an Undo command. The name of this command changes, depending on the specific action that it redoes.

SEE ALSO: *Undo*

Cut

Menu: Edit|Cut   **Hotkey:** <Ctrl>+<X>   **Toolbar:** Standard

The Cut commands removes any selected object or objects from your drawing and places them in the clipboard. From the clipboard you can paste them into *KeyCAD* drawings or into other Windows applications using the Paste or Paste Special commands.

SEE ALSO: *Copy , Paste, Paste Special, Copy Entities Tools*

Copy

Menu: Edit|Copy   **Hotkey:** <Ctrl>+<C>   **Toolbar:** Standard

The Copy commands copies any selected object or objects from your drawing and places them in the clipboard. From the clipboard you can paste them into *KeyCAD* drawings or into other Windows applications using the Paste or Paste Special commands.

SEE ALSO: *Cut, Paste, Paste Special, Copy Entities Tools*

Paste

Menu: Edit|Paste   **Hotkey:** <Ctrl>+<V>   **Toolbar:** Standard

The Paste command pastes into your drawing any data that has been copied or cut into the clipboard. You can paste *KeyCAD* drawing objects, as well as text and bitmaps, that have been placed in the clipboard in other programs.

Paste Special

Menu: Edit|Paste Special

The Paste Special command displays the Paste Special dialog.

Paste Special dialog

The Paste Special dialog lets you choose how to paste into your drawing an object that was previously cut or copied to the clipboard. The object on the clipboard can be from *KeyCAD*, or it can come from another application. The options available in the Paste Special dialog depend on the type of object in the clipboard, and on the program in which the object was created (the *source* application).

When you open the Paste Special dialog, you will see a number of choices in the As list box. For example, if there is a document on the clipboard created in Microsoft Word, the As text box will contain the following options:

- Word Document

- Picture

- Text

If you choose Word Document, the data in the clipboard will be inserted into your drawing as an OLE object, which you can edit within *KeyCAD* using Microsoft Word features. If you choose Picture, the data will be inserted as an image. If you choose Text, the data will be inserted as text, creating a *KeyCAD* text object, which you can treat exactly like text created with the *KeyCAD* Text tool.

If you choose to insert a clipboard object into your drawing as an OLE object, you have a choice of either *linking* or *embedding* the object. If you click the Paste option button, you will embed the OLE object in your drawing. This means that the drawing will have its own independent copy of the OLE object: Any changes you make to this object will *not* affect the file on which it was based. If you activate the Paste Link option button, the OLE object will be *linked* to the file on which it was

based. This means that any changes made to the file will be reflected in the OLE object, and any changes made to the OLE object will also affect the file.

If you check the Display as Icon check box, the OLE object will appear in your drawing as an icon. (When this box is checked, you can click on the Change Icon button to select another icon.)

NOTE: *There are only two types of data that can be inserted into KeyCAD as an editable KeyCAD drawing object. These are text data, which is inserted as a KeyCAD text object, and actual KeyCAD drawing entities that have been copied onto the clipboard from KeyCAD drawings. All other clipboard objects will be pasted either as OLE objects or as pictures.*

TIP: *If you plan to insert multiple copies of a single object into your drawing, you can keep your drawing file from growing too large by linking the objects rather than embedding them. This is possible because each linked object is simply a reference to the file on which it is based. If you link the object multiple times, each copy of the object shares the same source of data.*

SEE ALSO: *Edit Links Dialog, Edit Object , Insert Object , Insert Picture, Paste*

Clear Commands

Menu: Edit|Clear

The Clear commands let you remove graphic objects and other objects from a drawing. The Edit|Clear menu item opens a submenu containing four options: Selection, All, Construction, and All Constructions. You can reverse any clear with the Undo command.

The following table summarizes the Clear commands.

Command	Function
Clear Selection	Clear all objects that are currently selected.
Clear All	Clear all objects in the drawing.
Clear Construction	Activate the clear construction tool, with which you can delete individual constructions.
Clear All Constructions	Clear all construction entities in the drawing.

Clear Selection

Menu: Edit|Clear|Selection **Hotkey:** <Del>

Use the Clear Selection command to delete any graphics or other objects that have been selected. If no objects have been selected, then this command is unavailable.

SEE ALSO: *Select Commands, Cut*

Clear All

Menu: Edit|Clear|All

Use the Clear All command to delete everything on the drawing space and begin a new drawing.

Clear Construction

Menu: Edit|Clear|Construction

The Clear Construction command activates the Clear Construction tool, with which you can delete individual construction lines and circles. To use it, first activate the tool, then click on any construction that you wish to delete. Choose Cancel from the local menu or switch to another tool when you are finished.

Clear All Constructions

Menu: Edit|Clear|All Constructions

Use the Clear All Constructions command to delete all construction entities (construction lines and circles) from your drawing.

Select Commands

If you want to perform an editing action on an object or a set of objects in your drawing, you often need to select the objects first. When you select objects, they change to a pale magenta color and appear inside a frame with handles.

KeyCAD provides a large set of selection options, summarized in the following table.

Command	Function
Select	Activate the selection tool.
Select All	Select all objects in the drawing.
Select By	Access a submenu that lets you select objects by their type, layer, or attribute.

Select

Menu: Edit|Select   **Toolbar:** Insert

The Select tool lets you select and deselect objects in the drawing area using the mouse. You can select a graphic entity (as well as an OLE object or bitmap) by clicking on it, or you can select objects using a selection rectangle. Once objects are selected, you can deselect everything by clicking the mouse anywhere in the drawing area outside of the selection rectangle. You can also deselect everything by pressing <Esc> or choosing Cancel from the local menu.

WARNING: *If you try to deselect everything by clicking inside the selection rectangle, nothing will happen: You need to click outside the selection rectangle. Normally this is not a problem, but it might be confusing if you are zoomed in so that your view is* inside *the selection rectangle. If you find yourself in this situation just press <Esc> or choose Cancel from the*

local menu to deselect everything.

Selection by Clicking

To select any object, simply activate the Select tool, then click on the object.

To select a series of objects, adding them one at a time to the selection, hold down the <Shift> key while clicking on each object in succession. To deselect multiple objects one at a time, hold down the <Shift> key and click on each selected object in succession.

Selection within Rectangle

The selection rectangle gives you different results, depending on whether Open Window mode is on or off. To change this option, click the right mouse button in the drawing area to open the local menu, then choose Open Window mode. If the menu item has a check next to it, the option is on; otherwise the option is off.

To use the selection rectangle with the Open Window option on, hold the left mouse button down and move the cursor, creating a rectangle over an area that contains *any part* of the objects that you want to select, then release the mouse button. If any portion of an object is inside the rectangle, the entire object will be selected.

To use the selection rectangle with the Open Window option off, hold the left mouse button down and move the cursor, creating a rectangle over an area that contains the objects that you want to select *in their entirety,* then release the mouse button. If any portion of an object is outside the rectangle, it will not be selected.
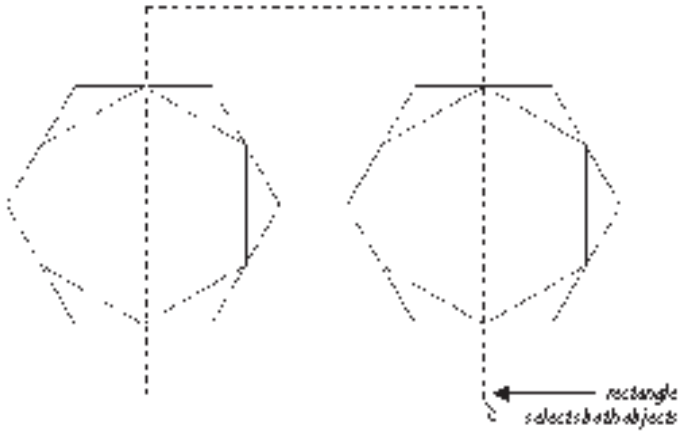
Selection rectangle with Open Window option on
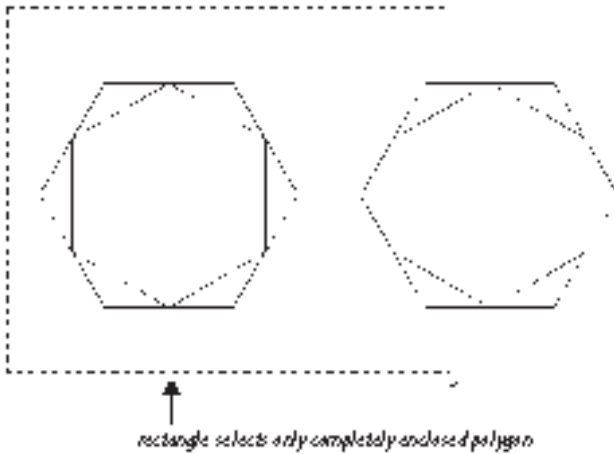
Selection rectangle with Open Window option off

Select All

Menu: Edit|Select All

Use the Select All command to select all objects that are currently on the drawing.

Select By Commands

Menu: Edit|Select By

Select By opens a submenu of special selection commands, summarized in the following table:

Command	Action
Entity Type	Brings up a dialog that lets you select objects by their type.
Layer	Brings up a dialog that lets you select all the object on a layer or subset of layers.
Attribute	Brings up a dialog that lets you select objects that share the same text in their Attribute property.

Select by Entity Type Dialog

Menu: Edit|Select By|Entity Type

The Select By Entity Type command opens a dialog that lets you select all the objects in the drawing that belong to specific types.

Select by Entity Type dialog

You can use this dialog to select objects that belong to any type or combination of types, using common Windows list box selection techniques. For example, if you wish to select all linear and angular dimensions in your drawing, you would hold down the <Ctrl> key while clicking on Linear Dimension and Angular Dimension, highlighting both

items. If you wish to add objects of these types to a current selection, place a check in the Add to the Current Selection check box. If this box is not checked, then any currently selected items will be deselected as the new objects are selected.

Select by Layer Dialog

Menu: Edit|Select By|Layer

The Select By Layer command selects all objects that exist on a specific layer or on a specific set of layers.

Select by Layer dialog

You can use this dialog to select objects that exist on a particular layer or combination of layers, using common Windows list box selection techniques. For example, if you wish to select all objects on layers Floor and Furniture (assuming that you have created these layers in your drawing), you would hold down the <Ctrl> key while clicking on Floor and Furniture, highlighting both items. If you wish to add objects of these types to a current selection, place a check in the Add to the Current Selection check box. If this box is not checked, then any currently selected items will be deselected as the new objects are selected.

SEE ALSO: *Layers Property Sheet*

Select by Attribute Dialog

Menu: Edit|Select By|Attribute

The Select by Attribute dialog lets you select all objects that share the same text in their Attribute property, which you can set for each object in the General property sheet of the Format|Properties dialog.

Select by Attribute dialog

To use this dialog, type text in the Attribute text box that matches the contents of the attribute text box of a set of entities, then click OK to select the entities. Because the attribute property contains group names, you can use this dialog to select all groups that share the same name. The dialog also contains the following options:

Case Sensitive. Place a check mark in this box if you want the search to distinguish between uppercase and lowercase letters. Entering **Wall** with this checked would select objects that have the attribute "Wall," but not "wall" or "WALL".

All Entities. Select this option button if you want to select all entities that share the specified text attribute.

Group Names Only. Select this option button if you want to select only groups that share the specified text attribute.

Text Entities Only. Select this option button if you want to select only text entities that share the specified text attribute.

NOTE: *The text attribute of a text entity contains the same text that the entity displays.*

You can also select objects that share attribute text that begins with the same characters by using the ***** symbol. The ***** symbol stands for any terminating characters in the text. For example, objects with text attributes "Wall 1," "Wall 2," and "Wall 3" will all be selected if you type **Wall *** into the Attribute text box. This is convenient for selecting all groups that belong to the same general category. You might want to keep this in mind when you are choosing names for groups.

NOTE: *All characters that follow the * symbol will be ignored. If you type* **Wall * with Window,** *for example, you will select objects with the attribute "Wall 1," and "Wall 2," as well as "Wall 1 with Window" and "Wall 2 with Window".*

SEE ALSO: *Properties Dialog*

Copy Entities Tools

Menu: Edit|Copy Entities

KeyCAD's tools for copying entities enable you to make single or multiple copies of entities, and at the same time precisely control the placement of the copies.

The following table summarizes the entity copying tools:

Command	Function
Linear Copy	Make copies along a line, specifying the interval between each copy.
Radial Copy	Arrange copies in an arc, specifying the angular distance between each copy and the rotation of each copy relative to the previous copy.
Array Copy	Copy entities in a two-dimensional array, specifying the distance between each entity.
Fit Linear Copy	Make copies along a line, specifying the total distance into which all the copies fit.
Fit Radial Copy	Arrange copies in an arc, specifying the total angular distance in which all the copies must fit and the rotation of each copy relative to the previous copy.
Fit Array Copy	Copy entities in a two-dimensional array, specifying the dimension of the rectangle into which they fit.
Mirror Copy	Create a mirror image of an object on the opposite side of a line that you specify.
Vector Copy	Make a single copy of an object, specifying the angle and distance of the copy from the original.

Tip: *When you are using the Copy Entities tools, you may often find it necessary to switch back and forth between the Edit Bar and the drawing area. One reason for this is to use the Edit Bar to specify the number of copies that you want to make. Remember that you can move to the Edit Bar without moving the mouse by pressing <Tab>. You can return to the drawing space from the Edit Bar, with the cursor in exactly the position you left it, by either clicking once in the drawing area or by pressing <Ctrl> + <D>.*

NOTE: *If you do not change the rotation or scale of the copies that you create, KeyCAD will include the original copy in the number of sets. If you do rotate or scale the copies, the original will* not *be included in the number of sets. For example, if you specify a new scale or rotation and specify 20 sets, KeyCAD will make 20 copies of the original. If you do not specify a new scale or rotation, KeyCAD will make 19 copies, using the original as the 20th copy.*

Linear Copy

Menu: Edit|Copy Entities|Linear   **Toolbar:** Copy

Use Linear Copy to create one or more copies of an object along a straight line, specifying the distance between each copy, the angle of the line along which the copies will be made, and the number of copies.

1. Select the object or objects that you want to copy.

2. Choose the Linear Copy tool.

3. Press <Tab> or click in the Edit Bar to activate it

4. Type these values into the Edit Bar fields:

   • Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.

   • Type values into XStep and YStep to specify the horizontal and vertical distance of each copy from the previous copy.

   • Type the number of copies you want to make into Sets.

5. Press <Enter> to create the copies.
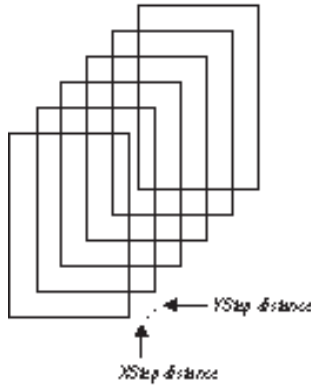
Linear copies

NOTE: *You can also specify the XStep and YStep values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of sets, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.*

Radial Copy

Menu: Edit|Copy Entities|Radial   **Toolbar:** Copy

Use Radial Copy to arrange one or more copies of an object in an arc, specifying the angular distance from one copy to the next.

1. Select the object or objects that you want to copy.

2. Choose the Radial Copy tool.

3. Define a point for the center point of the arc around which you want to arrange the copies. The radius of the arc is the distance from this point to the reference point of the selection. This arc is visible as a rubber-band circle in the drawing area.

4. Press <Tab> or click in Edit Bar to activate it, then enter the following values into the Edit Bar fields:

   • Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.

   • Type the number of copies you want to make into Sets. If you

have scaled or rotated the copies, the original will not be included in the number of sets. *KeyCAD* will choose appropriate default values for Angle and +Rot based on the number of sets you enter.

- Optionally type a value into the Angle text box to specify the angular distance between the copies.

- Optionally type a value into +Rot to specify the rotation of each copy relative to the previous copy.

5. Press <Enter> to create the copies.
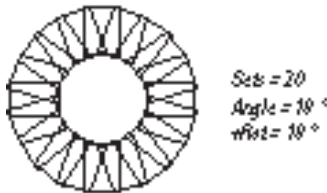
Radial copies

Notes: You can also specify Angle by defining a point within the rubber band circle in the drawing area using the mouse. However, the number of sets, the +Rot rotation value, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.

The Sets, Angle, and +Rot values are interdependent. Changing the number of sets will change the values in Angle and +Rot; changing Angle will affect +Rot. These default ratios are designed to give you an elegant, symmetrical radial copy. Once you have entered values in these fields, however, the automatic interdependency is switched off, and you can enter any values you like.

Array Copy

Menu: Edit|Copy Entities|Array   **Toolbar:** Copy

Use Array Copy to arrange copies of selected objects into a two-dimensional array (rows and columns). You specify the horizontal and vertical distance between the copies.

1. Select the object or objects that you want to copy.

2. Choose the Array Copy tool.

---

3. Press <Tab> or click in the Edit Bar to activate it, then enter the following values in the Edit Bar fields:

- Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.

- Type values in the XStep and YStep fields to specify the horizontal and vertical distance between the center points of the copies.

- Type values into the Rows and Columns fields to specify the number of rows and columns in the array.

4. Press <Enter> to create the copies.
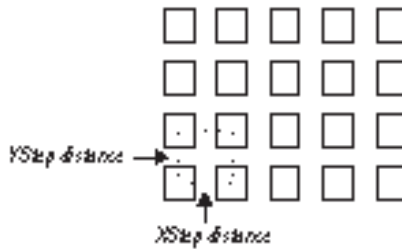
Array copies

NOTE: *You can also specify the XStep and YStep values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of rows and columns, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.*

Fit Linear Copy

Menu: Edit|Copy Entities|Fit Linear   **Toolbar:** Copy

Use Linear Fit Copy to create one or more copies of an object along a straight line, specifying a length into which all the copies will fit.

1. Select the object or objects that you want to copy.

2. Choose the Linear Copy tool.

3. Press <Tab> or click in the Edit Bar to activate it, then enter these values into the Edit Bar fields:

- Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.

- Type values into XSize and YSize to specify the length of the line onto which all the copies will fit. The copies will be placed on this line an equal distance apart.

- Type the number of copies you want to make into the Sets field. If you have scaled or rotated the copies, the original will be included in the number of sets.

4. Press <Enter> or create the copies.
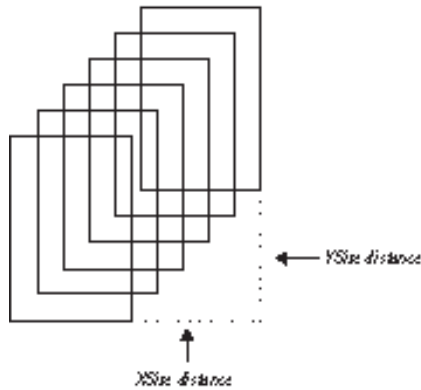
Linear fit copies

NOTE: *You can also specify the XSize and YSize values by defining a point relative to the selection in the drawing area using the mouse or Coordinate Fields. However, the number of sets, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.*

Fit Radial Copy

Menu: Edit|Copy Entities|Fit Radial   **Toolbar:** Copy

1. Select the object or objects that you want to copy.

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2.   Choose the Radial Fit Copy tool.

3.   Define a point for the center point of the arc around which you want to arrange the copies. The radius of the arc is the distance from this point to the reference point of the selection. This arc is visible as a rubber-band circle in the drawing area.

4.   Press <Tab> or click in Edit Bar to activate it, then type the following values in the Edit Bar fields:

•   Optionally type values into XScale, YScale, and Rotation if you want to scale and rotate at the same time as you copy.

•   Type the number of copies you want to make into the Sets field.

•   Optionally type a value into the Angle text box to specify the angular distance into which all the copies will fit.

•   Optionally type a value into the +Rot field to specify the rotation of each copy relative to the previous copy.

5.   Press <Enter> or click the mouse in the drawing area to create the copies.

KeyCAD will make radial copies in a counterclockwise direction, starting with the original object and ending at the angle set in step 4. The copy direction will be counterclockwise regardless of the direction setting in the Angle property sheet of the Drawing Setup dialog.
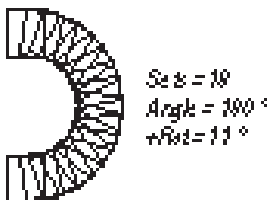
Fit radial copies

Notes: You can also specify Angle by defining a point within the rubber band circle in the drawing area using the mouse. However, the number of sets, the +Rot rotation value, as well as values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.

The Sets, Angle, and +Rot values are interdependent. Changing the number of sets will change the values in Angle and +Rot; and changing Angle will affect +Rot. These default ratios are designed to give you an elegant, symmetrical radial copy. Once you have entered values in these

*fields, however, the automatic interdependency is switched off, and you
can enter values into fields without affecting values in the related fields.*

Fit Array Copy

Menu: Edit|Copy Entities|Fit Array   **Toolbar:** Copy

Use Fit Array Copy to arrange copies of selected objects into a two-
dimensional array (rows and columns). The reference points of the
outermost objects in the array will align to the edges of a rectangle that
you define.

1. Select the object or objects that you want to copy.

2. Choose the Fit Array Copy tool.

3. Press <Tab> or click in the Edit Bar to activate it.

    •   Optionally type values into XScale, YScale, and Rotation if you
        want to scale and rotate at the same time as you copy.

    •   Type values in the XSize and YSize fields to specify the
        horizontal and vertical extents of the rectangle into which all the
        copies will fit.

    •   Type values into the Rows and Columns fields to specify the
        number of rows and columns in the array.

4. Press <Enter> to create the copies.
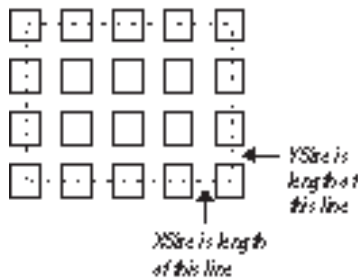
Fit array copies

NOTE: *You can also specify the XSize and YSize values by defining a point
relative to the selection in the drawing area using the mouse or
Coordinate Fields. However, the number of rows and columns, as well as*

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values for rotating and resizing the copies, must be specified in the Edit Bar first, unless you want to accept the current values.

Mirror Copy

Menu: Edit|Copy Entities|Mirror   **Toolbar:** Copy

Use Mirror Copy to create a mirror image of an object on the opposite side of a line that you specify.

1. Select the object or objects that you want to copy. If the Edit Bar is not visible, display it using View|Toolbars.

2. Choose the Mirror Copy tool.

3. Click in the drawing area or use the Coordinate Fields to define a point in the drawing area through which a mirror construction line will pass.

4. Enter the angle of the mirror construction line in the Angle field, and press <Enter>. Alternatively, you can use the Coordinate Fields or click in the drawing area to define a second point through which the mirror construction line will pass.

KeyCAD will draw a mirror image of the selected objects at the same distance from the line on the opposite side.

Vector Copy

Menu: Edit|Copy Entities|Vector   **Toolbar:** Copy

Use Vector Copy to place a copy of an object at a specified angle and distance from the reference point of the original.

1. Select the object or objects that you want to copy.

2. Choose the Vector Copy tool.

3. Use the Coordinate Fields or click in the drawing area to define two points of a virtual line to specify the angle and distance of the copy from the original. This line appears in the drawing area as an arrow.

Alternatively, you can define the angle and distance of the vector directly in the Edit Bar and press <Enter>.
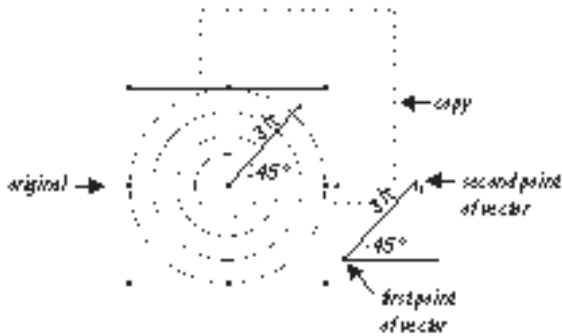
Vector copy

Edit Links

Menu: Edit|Links

Edit Links displays a dialog in which you can make changes to any linked OLE objects in the current drawing. OLE objects are created in other Windows applications and inserted into your *KeyCAD* drawing with the Insert Object command. You can edit an OLE object within *KeyCAD* using the Windows application in which the object was created (the *source* application). For example, you can insert a Windows Paint object into your drawing and edit it within *KeyCAD* using the Windows Paint tools.

A *linked* object is one that retains its link to the file on which it is based. Changes to the OLE object affect the file, and changes to the file affect the object. These changes take place when the OLE object is *updated*. You can set each linked object so that the update happens automatically, or so that the update must be done manually.

Links dialog

The Links list box contains all the links in the active drawing. The changes that you make in the Edit Links dialog affect any links that are highlighted in the list. You can highlight multiple links by holding down the <Shift> or <Ctrl> key and clicking in the list. The Links column in the list gives you the name and path of the file to which the OLE object is linked. The Type column tells you the name of the source application. The Update column tells you whether the link is set to be updated automatically or manually.

Update Option. These option buttons enable you to set whether the selected links are updated automatically or manually.

Update Now. This button updates the current link immediately. This is how you update a link if it is set to be manually updated.

Open Source. This button opens the source file in the application that created it. This enables you to compare the linked object to the source file, and it allows you to edit the file in the source application.

Change Source. This button opens a dialog that lets you choose a new source for the linked OLE object. Once the OLE object is linked to the new file, its contents will change to match the new file.

Break Link. This button terminates the link between the OLE object and its source file. Once the link is broken, the OLE object is treated as an *embedded* object. It is still an OLE object, but changes to it no longer affect the source file, and changes to the source file no longer can affect the object. Breaking the link also affects the size (in bytes) of your drawing, since the embedded object contains its own copy of the data in the object, while the linked object contains only a reference to the object.

SEE ALSO: *Edit Object, Embedded and Linked OLE Objects, Insert Object*

Edit Object

Menu: Edit|Object   *KeyCAD for Windows*

5

Drawing Environment

View Menu and Tools

The View menu provides features that let you control two aspects of the drawing environment. These features enable you to:

- Change your view of the drawing area by zooming and panning.
- Set up your desktop by displaying and hiding toolbars and other interface controls.

The first part of this chapter discusses some common techniques and concepts related to views and to setting up the *KeyCAD* interface. The remainder of the chapter provides a complete reference to the commands and tools available from the View menu.

Views of the Drawing Space

The *KeyCAD* drawing space is a plane on which you can place drawing entities and other objects. A *view* is the area of the drawing space that appears on your screen. As your drawing becomes larger and more complex, it's important to be able to find views that let you work at the right location in your drawing, at an appropriate level of detail. You may need to concentrate your work within a tiny portion of the drawing, you may need to pull back and see all the objects on the drawing, or you may need to see the drawing at the size it will be when printed. *KeyCAD* offers a variety of options for panning and zooming that get you quickly to the exact view that you need. If you arrive at a view that you want to return to later, you can save it as a Named View, then return to that view at any time.

Zooming

Zooming means to move in or out of the drawing space, viewing the drawing at a greater or lesser level of detail. The View menu provides a number of zooming commands, which will be discussed in detail in the Command Reference section of this chapter.

One quick and easy way to zoom is to use the plus and minus keys of the numeric keypad: plus zooms in one step; minus zooms out one step. The amount that you zoom in each step depends on the *zoom factor*, which by default is set to 2. You can set the zoom factor in the Tools|Program Setup dialog.
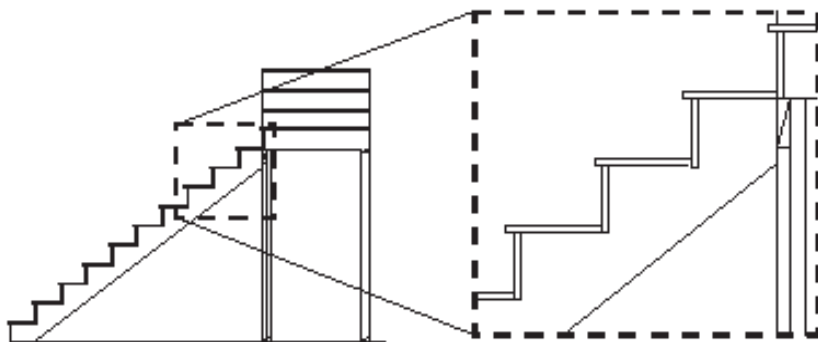
Zooming in on a view

NOTES: *When you zoom in and out using the plus and minus keys, the view centers on the mouse cursor. If you zoom in using the menu command, you will zoom in on the center of the current view, regardless of the position of the mouse cursor.*

As you zoom out, KeyCAD simplifies the representation of very small objects on the screen to increase redraw speed. You can set the size at which this occurs using the Suppress Display... control on the Tools/Display property sheet.

TIP: *To zoom in on a particular point in the drawing and place it at the center of the screen, place the mouse cursor over the point and press the <+> key on the numeric keypad.*

SEE ALSO: *Display Property Sheet, Program Setup Dialog, Zoom Commands*

Panning

Panning means to move the view to another location across the plane of the drawing. *KeyCAD* provides several different options for panning, which will be discussed in the Command Reference section of this chapter.

Two simple ways of panning are to use the scroll bars or arrow keys. The scroll bars work the same way that they do in other Windows programs. You can click the scrollbar arrow buttons to scroll one step, click in the body of the scrollbar to scroll about one third of a screen, or drag the "thumb" of the scrollbar to move quickly across the drawing. The arrow keys are also simple and intuitive: press the up, down, left, or right arrow key to pan a single step.

view before pan →   ← *view after pan*
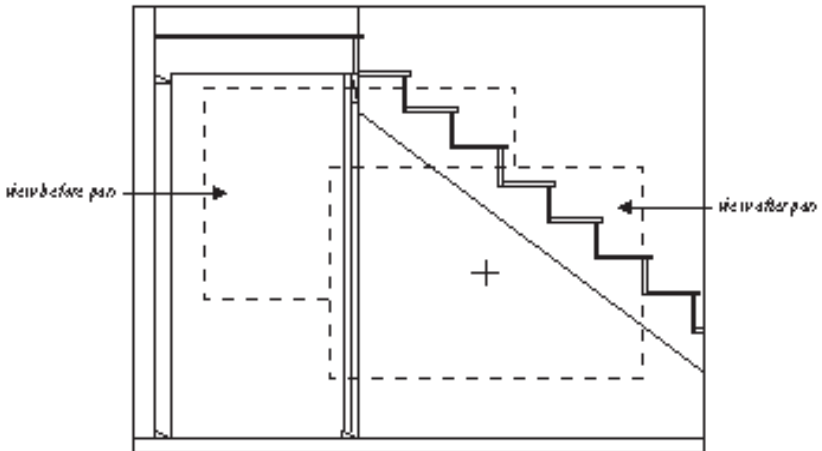
Panning to a view

TIP: *To quickly pan to a specific point, place the mouse cursor at the point, then type <Ctrl>+<End>. You will instantly move to a view with the point you chose at the center of the screen.*

Setting Up the Desktop

In *KeyCAD* 3.0 you can set up the user interface to suit your own preferences, or to fit the requirements of a particular drawing task. You do this by selectively displaying and hiding toolbars and other interface controls, and by positioning or "docking" toolbars and other movable controls, such as palettes, on the *KeyCAD* desktop.

TIP: *You can control whether KeyCAD saves your desktop arrangement when you exit by setting the Save Desktop on Exit option in the Program Setup dialog.*

SEE ALSO: *Program Setup Dialog, KeyCAD Interface Tutorial*

Displaying and Hiding Controls

The View menu provides a number of commands for displaying and hiding interface controls such as the Edit Bar, Status Bar, and rulers. Each of these commands is a *toggle*, which means that it acts as an on/off switch. If the menu command is displayed with a check mark, the control is currently displayed. If the menu command is displayed without a check mark, the control is hidden.

You can display and hide the *KeyCAD* toolbars through the Tools|Desktop property sheet, which is also accessible through the View|Toolbars command.

TIP: *It's generally a good idea to set your preferred settings in the Tools|Desktop property sheet, and then use the View menu to show and hide individual controls as you are working.*

SEE ALSO: *Desktop Property Sheet*

Docking Toolbars

You can move any toolbar, or the Edit Bar, by positioning the mouse over its edge and dragging it to another position. There are three places on the *KeyCAD* screen where you can "dock" toolbars: the upper toolbar area, just below the menu bar; the left or right edge of the screen, which expands to form a vertical docking area; and the drawing area. When you dock a toolbar on the upper part of the screen it will be oriented horizontally. If you place it along the left or right edge of the screen it will be positioned vertically. Toolbars placed over the drawing area will be contained in their own window, which acts as a floating palette. A floating palette is always visible in front of your drawing.

NOTE: *The Edit Bar and the toolbars that contain text fields (Property and Text Format), cannot be positioned vertically.*
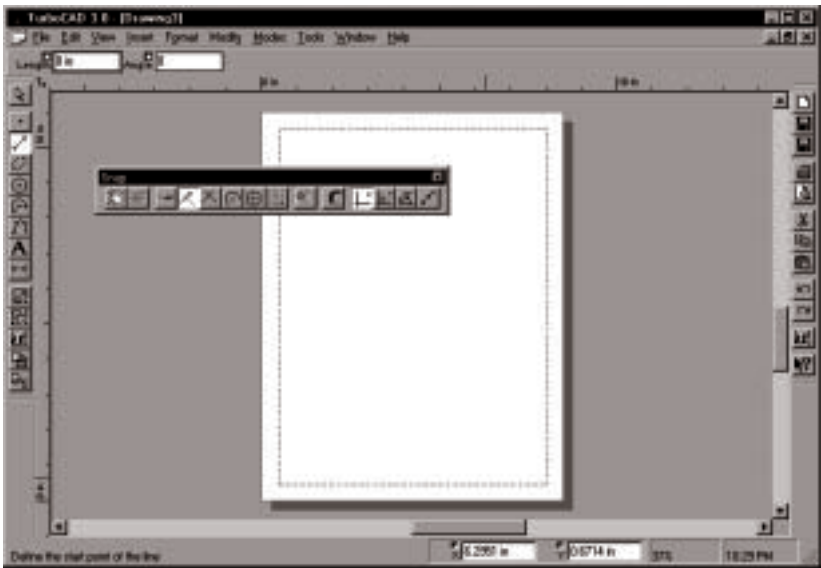
Four places to put toolbars

Command Reference

The following table summarizes the commands you can access from the View menu:

Command	Function
Redraw	Manually refresh a drawing that has changed.
Zoom	Access tools and commands for moving your view closer to or further away from an area of the drawing.
Pan to Point	Pick a point in the drawing and move it to the center of the screen.
Vector Pan	Pick a point in the drawing and move it to a particular location on the screen.
View Grid	Display or hide the grid.
View Crosshairs	Display crosshairs in place of mouse cursor.
View Aperture	Change cursor to a circle that represents the snap aperture.
View Rulers	Display or hide the rulers.
View Status Bar	Display or hide the Status Bar.
View Edit Bar	Display or hide the Edit Bar.
View Symbol Library	Display or hide the Symbol Library palette.
View Toolbars	Display the Desktop property sheet in which you can display or hide toolbars.
View Selection Info	Display or hide the information about the currently selected object.

Redraw

Menu: View|Redraw   **Hotkey:** <F5>   **Toolbar:** View

In most cases *KeyCAD* will automatically redraw your screen whenever the drawing changes. Occasionally, however, you may find that the screen is not completely "cleaned up" after a drawing or editing operation. If this happens, you can manually redraw the screen using the Redraw command.

Zoom Commands

Menu: View|Zoom   **Toolbar:** View

The Zoom commands and tools let you "zoom" into your drawing to view detail areas close up, or pull back to view larger areas of your drawing. The following table summarizes the Zoom commands and tools:

Command	Action
Zoom In	Move one step into the drawing.
Zoom Out	Move one step further from the drawing.
Zoom Window	Activate the Zoom Window tool, with which you can select an area to view.
View Extents	Zoom to a view that contains all of the objects in the drawing.
Full View	Return to "starting" view of the drawing.
View Printed Size	View the drawing at the size it will be when printed.

Zoom In

Menu: View|Zoom|Zoom In   **Hotkey:** <Gray +>   **Toolbar:** View

The Zoom In command magnifies the current view of the screen. The scale of magnification is determined by the *zoom factor*, which by default is equal to two. You can reset the zoom factor in the General property sheet of the Tools|Program Setup dialog.

NOTE: *If you zoom in using the menu command, you will zoom in on an area centered on the current view. If you zoom in using the <Gray +> key, you will zoom in to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.*

SEE ALSO: *Program Setup Dialog*

Zoom Out

Menu: View|Zoom|Zoom Out **Hotkey:** <Gray -> **Toolbar:** View

The Zoom Out command backs away from the area where the mouse cursor is currently located. The amount that the view pulls back in each step is determined by the *zoom factor*, which by default is equal to two. You can reset the zoom factor in the General property sheet of the Tools|Program Options dialog.

NOTE:*If you zoom out using the menu command, you will zoom out to an area centered on the current view. If you zoom out using the <Gray +> key, you will zoom out to an area defined so that the mouse cursor is kept in the same relative position within the view rectangle.*

SEE ALSO: *Program Options Dialog*

Zoom Window

Menu: View|Zoom|Zoom Window   **Hotkey:** <Ctrl>+<Home>
Toolbar: View

The Zoom Window tool enables you to select a rectangle area of the screen and zoom in to a view of the that area. To use this tool:

1. Activate the Zoom Window tool.

2. Click to specify one corner of the zoom window.

3. Move the mouse cursor to the point where you want to place the diagonally opposite corner of the zoom window, so that the zoom window encompasses the area that you want to view.

4. Click to specify the opposite corner of the window. *KeyCAD* will immediately zoom in on the area that the zoom window encloses.

You can also drag the mouse to create the zoom window, in which case your view will zoom in immediately after you release the mouse button.

NOTE: *Because the dimensions of the zoom rectangle will probably not be exactly proportional to the screen, KeyCAD chooses the closest view based on the zoom rectangle you define.*

View Extents

Menu: View|Zoom|Extents   **Hotkey:** <Ctrl>+<Backspace>
Toolbar: View

The View Extents command zooms to a view of the smallest area of your
drawing that completely contains all of the objects in your drawing. This
is the same as the area enclosed by the selection rectangle if you select
all the objects in the drawing.

Full View

Menu: View|Zoom|Full View   **Hotkey:** <Shift>+<Backspace>
Toolbar: View

The Full View command returns you to the default view at which you
began your drawing.

View Printed Size

Menu: View|Zoom|Printed Size   **Toolbar:** View

The View Printed Size command takes you to a view of the drawing at
approximately the size it will be when printed. In this view one inch of
screen space is approximately equal to one inch on paper.

NOTE: *The exact correspondence of screen space to paper space will vary,
depending on your graphics display and screen resolution.*

Pan Commands

The Pan commands and tools move your view across the drawing area, in
any direction. The following table summarizes the Pan commands and tools.

Command	Function
Pan to Point	Choose a point and scroll the drawing so that the point is at the center of the screen.
Vector Pan	Choose a point and a destination and scroll the drawing so that the point moves to the destination.

Pan to Point

Menu: View|Pan to Point   **Hotkey:** <Ctrl>+<End>

Use the Pan to Point feature to choose a point on the screen and scroll the
drawing so that the point you chose moves to the center of the screen.
This feature is available from the menu as a tool and from the keyboard
as a command. To use the menu version of the feature:

1.   Choose View|Pan to Point to activate the Pan to Point tool. The
     cursor will change to an arrow pointing to the center of a small
     cross-hair icon.

2.   Click on any point in the drawing. This point will move to the
     center of the screen.

To use the keyboard Pan to Point command, place the cursor at a point on
the screen and press <Ctrl>+<End>. The point at the location of the
cursor will scroll to the center of the screen.

Vector Pan

Menu: View|Vector Pan

Use the Vector Pan tool to choose a point and a destination, and scroll the
drawing so that the point moves to the destination. To use Vector Pan:

1.   Activate the Vector Pan tool.

2.   Click at any location in the drawing area. This specifies the
     location of the point that you would like to move (the *base point*).

3.   Move the cursor to the destination point and click. The drawing
     will scroll so that the base point moves to this destination.
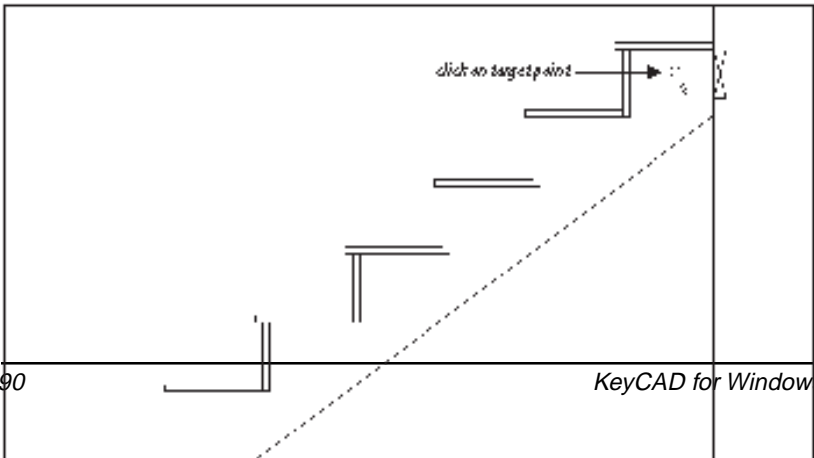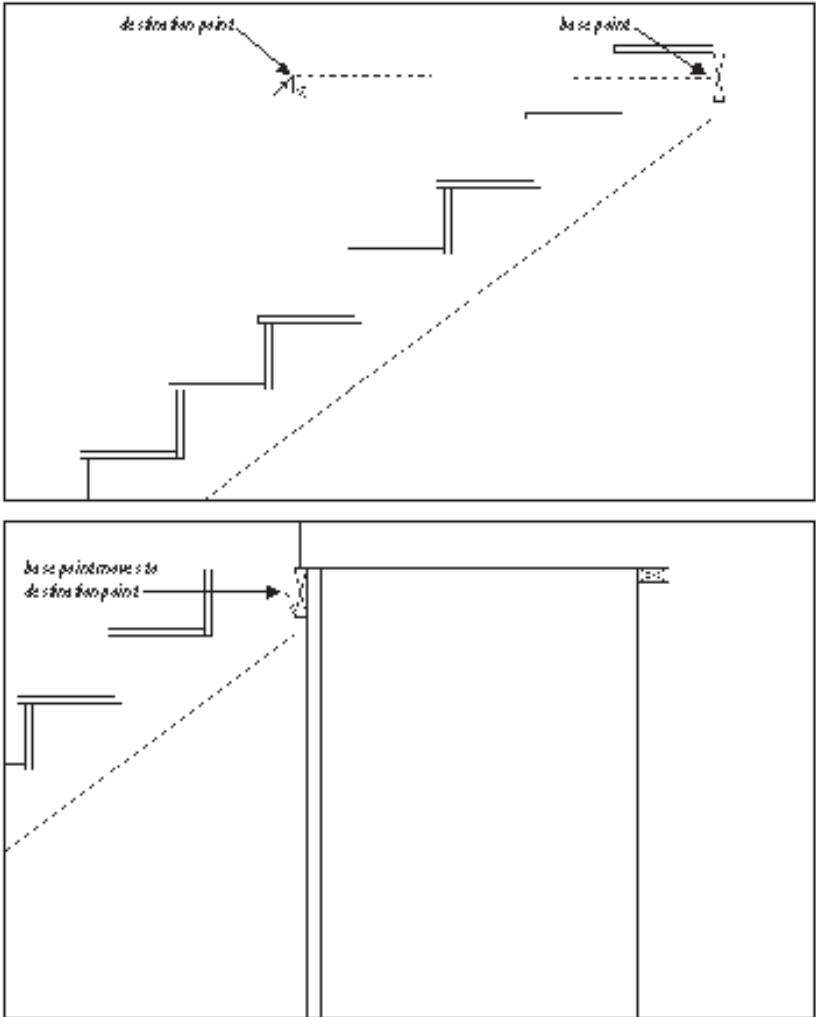
Using Vector Pan

View Grid

Menu: View|Grid

The View Grid command is a toggle that displays and hides the grid. The grid is a pattern of dots, crosses, or lines that divides your drawing space geometrically into Cartesian coordinate locations. You can define points at positions on the grid using the Snap to Grid snap mode. You can set

the spacing, type, and display parameters of the grid in the Tools|Grid property sheet.

NOTE: *Making the grid visible or invisible does not affect whether the grid is active. The grid can be visible and inactive, and it can be invisible and active. To control whether the grid is active, use the Snap to Grid command from the Modes|Snaps menu, the Snaps toolbar, or the local menu.*

SEE ALSO: *Coordinate Systems, Grid Property Sheet, Snap Modes Grid*

View Aperture

Menu: View|Aperture

The View Aperture command is a toggle that displays the snap aperture. The snap aperture is the circular area in which an entity needs to be located in order to snap to a point located on the entity, or in which a grid point or graphic point must be located in order to snap to that point.

TIP: *You can set the radius of the snap aperture in the Tools|Program Setup dialog.*

SEE ALSO: *Snap Aperture, View Crosshairs*

View Crosshairs

Menu: View|Crosshairs

The View Crosshairs command is a toggle that changes the cursor to crosshairs and back to a normal cursor. The crosshair extension lines let you see more clearly how the current location of the cursor corresponds to positions on the ruler and on graphic objects.
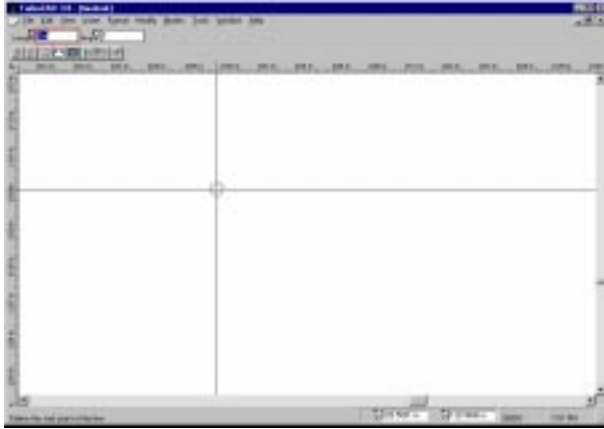
Crosshairs and snap aperture give you increased visual feedback

SEE ALSO: *Program Setup Dialog, View Aperture*

View Rulers

Menu: View|Rulers

The View Rulers command is a toggle that displays and hides the Rulers. Rulers are displayed along the top and left edges of the screen and show the measurements of your drawing in Page or World units. You can change between Page and World units by clicking the Paper/World button located near the upper-left corner of the drawing area at the intersection of the horizontal and vertical rulers.

You can change the units and scale of the drawing in the Units and Scale property sheet of the Tools|Drawing Setup dialog. The ruler will change to reflect the current scale and display.

SEE ALSO: *Drawing Setup Dialog*

View Status Bar

Menu: View|Status Bar

The View Status Bar command is a toggle that displays and hides the Status Bar. The Status Bar is displayed along the bottom of your screen.

It contains a brief status message, the Coordinate Fields, and a zoom percentage box on its right side, which displays the zoom percentage of your current view. At printed size, the drawing's zoom percentage is 100.

View Edit Bar

Menu: View|Edit Bar

The View Edit Bar command is a toggle that displays and hides the Edit Bar. The Edit Bar is a control that enables you to create and edit entities by entering distances, angles, and other data directly into the Edit Bar fields. You can drag the Edit Bar and place it anywhere in the tool area under the menu bar. You can also place the Edit Bar over the drawing space, where it will float over the drawing as a palette.

TIP: *Although you can draw in KeyCAD using only the mouse, many drawing and editing tools are more accurate if you work in the Edit Bar, and some tools have limited use without the Edit Bar. You will probably want to keep the Edit Bar on screen at all times.*

NOTE: *The fields in the Edit Bar change dynamically, depending on the currently active drawing or editing tool. The number of fields can vary quite a lot, so it's a good idea to place the Edit Bar toward the left edge of the screen so that all the fields will always be visible.*

SEE ALSO: *Drawing with the Edit Bar*

View Symbol Library

Menu: View|Symbol Library

The View Symbol Library command is a toggle that displays and hides the Symbol Library palette. The Symbol Library palette shows the symbols in the current symbol library.

Symbol Library palette

Symbols are actually fully independent *KeyCAD* drawings. A *KeyCAD* 3.0 symbol library is simply a Windows folder that contains any number of *KeyCAD* files. The Symbol palette displays the contents of a symbol library as large icons, small icons, or file names. If you click on any of the symbols displayed in the palette, a description of the symbol will appear in the description area at the bottom of the palette, as shown in the preceding illustration.

The title and description of the symbol is determined by information in the Summary Info dialog. The Title field of the Summary Info dialog determines the text that appears under the icon. The Subject field determines the icon description.

From the Symbol palette, you can:

- Place a symbol in your drawing.
- Turn a selected part of your drawing into a symbol.
- Change to another open symbol library.
- Open a new symbol library.
- Change your view of the Symbol Library.

The following sections cover these options.

SEE ALSO: *Symbol Libraries Property Sheet, Summary Info Dialog*

Placing a Symbol in a Drawing

To place a symbol in your drawing, simply drag it out of the Symbol palette and drop it in your drawing. The symbol will be placed in the drawing at the point where you release it. The symbol will remain

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selected after you place it, so that it can be repositioned and scaled.

The symbols that The Learning Company provides with *KeyCAD* are groups. This means that they have names, which makes them easier to keep track of, and it means that they can be selected and manipulated as a single unit.

WARNING: *Don't expect all symbols to be this well-behaved. A symbol is really just a KeyCAD drawing, and it won't be a group unless the person who created the symbol saved it as a group.*

Turning a Selection into a Symbol

Any *KeyCAD* drawing can be a symbol. To create your own symbol library, all you really need to do is save all the drawings that comprise the library into a single Windows folder. However, another equally good way to save symbols is to drag selections from drawings into the Symbol palette. This gives you the additional option of saving only part of an existing drawing as a symbol.

1. Select the parts of your drawing that you want to turn into a symbol. (If you want to drag your entire drawing into the Symbol palette, choose
    Edit|Select All.)

2. Before dragging it into the Symbol palette, it's a good idea to group the selection by choosing Format|Create Group. (This isn't strictly necessary, but it will make the symbol easier to work with later.)

3. Hold the mouse over the selection's reference point (the dot in the middle of the selection). You will know the mouse is in the right position when the cursor turns into a four-directional pointer.

4. Hold the mouse button down and drag the selection into the Symbol palette. Release the mouse button to drop the selection. The Save Symbol As dialog appears.

5. Save the symbol as a file, just as you would in the regular Save As dialog. You have the option of navigating to another folder, which will have the effect of placing the symbol into a different symbol library. If you put it in a folder that isn't already a symbol library, *KeyCAD* will open a new symbol library in your drawing.

SEE ALSO: *Save As Dialog*

Changing to Another Open Symbol Library

You can have multiple symbol libraries open in your drawing. (You open and name symbol libraries in the Tools|Symbol Library property sheet.) To view the open libraries from the Symbol palette, click the Options button to display the palette's local menu. At the bottom of the local

menu all of the open symbol libraries are displayed. To switch to another library, click on its name in the local menu. The contents of that library will then be displayed in the Symbol palette.

Opening a New Symbol Library

To open a new symbol library, click the Options button to display the palette's local menu and choose Open. This displays the Symbol Libraries property sheet, in which you can name and open new libraries.

SEE ALSO: *Symbol Libraries Property Sheet*

Changing Your View of a Symbol Library

The Symbol palette can display its symbols as icons, small icons, or as names. To switch between these three options, first click the Options button to display the palette's local menu, then choose Names, Small Icons, or Large Icons. A check mark will appear next to the currently selected option.

View Toolbars

Menu: View|Toolbars

The View Toolbars command displays the Desktop property sheet, which enables you to hide and view *KeyCAD* toolbars, as well as other interface controls.

SEE ALSO: *Desktop Property Sheet*

View Selection Info

Menu: View|Selection Info

The View Selection Info command displays and hides the Selection Info palette. This palette contains information about the currently selected object, such as its entity type, dimensions, and location in the drawing.

The Selection Info palette

The Selection Info palette contains two windows. The upper window shows the type and structure of the objects selected. If an object such as a group contains other objects, this relationship will be displayed in outline form. A plus sign next to an item indicates that it contains other items that are not displayed. To open the outline, and see what objects are contained, click on the plus sign; click again to close the outline to see a higher level of the hierarchy.

The lower window displays precise data about whatever object is currently selected in the upper window. It shows the object's graphic type, its coordinate location in the drawing space, and its dimensions in current World units.

Pressing the Copy button copies this data to the Windows clipboard so that you can paste it into a word processor or text editor, such as Windows notepad. The text that you copy contains special markers so that programmers, or very adventurous users, can "parse" the data in their own programs.

NOTE: *Experienced Web users may notice that the markers are similar to those used in HTML.*

6

Creating and Inserting Objects

Insert Menu and Tools

The Insert menu gives you access to all of the *KeyCAD* drawing tools, as well as to dialogs that let you insert various types of objects into your drawing. You can use the drawing tools to create:

- Drawing entities (such as lines, circles, and polygons).
- Construction entities (temporary lines and circles that you use as reference points or to mark off areas in your drawing).

You can also insert the following types of objects into your drawing:

- Files (all standard CAD file types).
- Pictures.
- OLE objects, which you can edit using other Windows applications.

NOTE: *There are, in addition, two other types of objects that can be inserted into a drawing symbols. You insert these directly from the Symbols palettes which you can display from the View menu.*

SEE ALSO: *View Symbol Library*

This chapter first discusses some common techniques used in drawing entities. The remainder of the chapter provides a complete reference to the commands available from the Insert menu.

Creating Entities

To use any drawing tool you need to understand a set of common techniques. These techniques include:

- Formatting objects by setting their properties.
- Defining points by using the mouse in combination with snap modes, or by using the Coordinate Fields.
- Specifying quantities, such as length and angle, using the Edit Bar.
- Finishing and canceling operations.
- Using the local menu.

The following sections describe these techniques. For a more detailed discussion of how to apply the techniques in the process of creating an actual drawing, run the *KeyCAD* online tutorial by choosing Tutorial from the Help menu.

Formatting Entities

When you draw an entity in *KeyCAD*, the entity is formatted using the current default properties of the tool with which you draw it. These properties include, among many other things, the width, style, and color of the pen, as well as the pattern and color of the brush. In addition to these common properties, some types of entities, such as text, double lines, curves, and dimensions, have properties that belong only to the specific entity type. To set the properties for a drawing tool, you can activate the tool and choose Format|Properties, or you can simply right-click on the drawing tool.

Once an entity is drawn, you can format it by changing its properties. To do this, first select the entity, then choose Format|Properties; or simply activate the Select tool, then double-click on the entity that you want to format (or select the entity, then choose Format|Properties).

NOTE: *You cannot access or change the properties of Construction Entities (construction lines and circles).*

SEE ALSO: *Setting Properties, Property Toolbar, Properties Dialog*

Defining Points and Specifying Measurements

To create any entity you must use some combination of the following three methods:

- Define points by clicking the mouse in the drawing area, using snap modes.
- Define points by entering their coordinates in the Coordinate Fields.
- Specify measurements (lengths, angles, and sometimes other information) in the Edit Bar.

Almost always, the first step in drawing any entity is to define its starting point. After the first point is defined, you create the entity by defining a series of points and/or specifying measurements.

For example, to draw a circle using the Center and Point tool, you first need to define a point for the center of the circle. After this point is defined, you have three ways of completing the circle:

- Define any point that will be on the perimeter of the circle using the Coordinate Fields or mouse.
- Specify the radius of the circle in the Edit Bar.
- Specify the circumference of the circle in the Edit Bar.

The method you choose depends on the specific characteristics of the tool you are using, the drawing task, and your own preferences.

WARNING: *Trying to draw an entity in the Edit Bar without defining the entity's starting point can sometimes lead to confusing results. It's almost always simpler to start drawing an entity by defining its starting point.*

Defining Points with the Mouse

In a sense, you define points with the mouse by simply pointing and clicking. However, what exactly happens when you define a point this way depends on *snap modes* and on *KeyCAD*'s *Ortho* setting. The following sections discuss these features briefly and point you to other sections of the manual where you can find out more.

Using Snap Modes

The result you get when you click the mouse in the drawing area depends, in part, on the current snap mode. If your snap mode is currently set to No Snap, then the point will simply be defined at the

position where you clicked the mouse. Otherwise, the location of the point will be calculated relative to a drawing object or to the grid. If there is no point to snap to based on the mouse position, *KeyCAD* will beep, and it won't define a point at all. To find out how to change snap modes, see the section "Setting Snap Modes and Ortho" below.

SEE ALSO: *Snap Modes*

Constraining Angles with Ortho

To constrain angles when you are drawing with the mouse, use Ortho mode. You can activate Ortho through the Modes menu, the Snap toolbar, or the local menu. The local menu turns Ortho on or off only until the next point is defined; *KeyCAD* then reverts to the Ortho setting in the Modes menu.

If you are using *KeyCAD*'s factory settings, Ortho restricts lines to orthogonal (vertical and horizontal) angles. You can configure Ortho mode to less restrictive settings in the Angle property sheet (choose Tools|Angle). Some CAD users, for example, prefer to restrict angles to increments of 15 degrees.

SEE ALSO: *Angle Property Sheet, Ortho*

TIP: *Another convenient way to turn Ortho on or off temporarily is to hold down the <Shift> key while drawing.*

Setting Snap Modes and Ortho

You can change snap modes and Ortho mode any time, even while you are in the middle of drawing an entity. For example, to create a spoke on a wheel you could snap the first point of a line to the center of a circle, then snap the second point of the line to a point on the perimeter of the circle. To set a default snap mode, use the Snaps toolbar or the main menu; to set a "one shot" snap mode that stays in effect until you define the next point, use the local menu. To use the local menu, click the right mouse button in the drawing area to open the menu, choose the desired snap mode, then click the mouse to define the point.

NOTE: *The local menu gives you "one shot" mode changes. If you change modes using the local menu, the mode will change back to the setting in the Modes menu after you define the next point.*

Defining Points Using the Coordinate Fields

In addition to using the mouse with snap modes and Ortho, you can also define points by entering them directly into the Coordinate Fields and then pressing <Enter>. Points defined in the Coordinate Fields are always defined at the exact coordinate positions specified, regardless of the snap mode. The type of coordinates you type in the Coordinate Fields depends on the coordinate system you are using. In the absolute and relative coordinate systems you use X,Y coordinates.

You can lock down the coordinates in the Coordinate Fields, so that the values in the boxes don't change when you move the mouse cursor over the drawing area. To do this, check their lockdown boxes (these are immediately to the left of the fields).

SEE ALSO: *Coordinate Fields, Coordinate Systems*

Tip: *The Coordinate Field lockdown boxes are particularly useful when you lock down just one dimension. If you enter a value in just one of the fields and lock it down, you can then set the second coordinate using the mouse. This makes it easy to define a series of points along a specific horizontal or vertical line.*

You can define points by entering coordinates in the Coordinate Fields

Drawing with the Edit Bar

The Edit Bar lets you draw entities *parametrically* by specifying dimensions and angles. To do this, you must define a starting point for the entity that you are drawing. For example, to draw a regular polygon using the Edit Bar, define the centerpoint of the polygon using the mouse or Coordinate Fields, and define the polygon's number of sides, radius, and start angle in the Edit Bar.

You can specify angles and lengths in the Edit Bar

NOTE: *Creating or editing an entity with the Edit Bar ignores Ortho and snap mode settings. The entity will be defined using the exact values that you enter in the Edit Bar.*

Using the Edit Bar Lockdown Boxes

Many fields on the Edit Bar have lockdown check boxes, similar to those on the Coordinate Fields. When these boxes are checked, the values in the fields don't change when you move the mouse over the drawing area. This makes it possible to lock down one of the values and manipulate the second value with the mouse.

For example, you might want to specify the exact radius of an arc in the Edit Bar, then use the mouse to determine the locations of the arc's starting and ending angles. After defining the starting point for the arc, you could type its radius into the Edit Bar, then lock down the Radius field. This prevents the radius from changing when you re-enter the drawing area. You could then use the mouse and snap modes to determine the arc's starting and ending angles.

Finishing and Canceling

Some entities are defined by a specific number of points.

Other entities, such as multilines and concentric circles, are composed of an indeterminate number of parts. You can add as many segments to a multiline, or as many circles to a set of concentric circles, as you like.

Users of earlier versions of *KeyCAD* may be confused by the way you cancel and finish such entities in *KeyCAD* 3.0. However, with a little practice you will find that *KeyCAD* 3.0 is consistent and intuitive in the way it handles these operations. It will help the transition if you remember these rules:

Cancel always means to stop the current drawing operation and start over. It removes the entire entity you are currently drawing. For example, if you press <Esc> or choose Cancel from the local menu while you are drawing a multiline, the entire multiline will vanish, not just the last segment.

Finish means to complete the entire entity. For example, if you are creating a multiline you can finish the entity by (1) double-clicking, (2) pressing the right mouse button and choosing Finish from the local menu, or (3) pressing <Ctrl>+<F> while working in the Edit Bar.

Pressing <Enter> in the Edit Bar means to accept the current setting in the Edit Bar fields. If you press <Enter> while drawing a multiline, the settings in the Edit Bar will be used to draw the next segment of the multiline. If you are creating an entity with a fixed number of steps, then pressing <Enter> in the Edit Bar will finish the entity, provided you have already defined a starting point.

NOTE: *When you are working in the Coordinate Fields, pressing <Enter> simply defines a point as specified in the Coordinate Fields.*

TIP: *Entities that can be closed geometrically, specifically the Multiline, Double Multiline, Bezier Curve, and Spline Curve tools provide another option for finishing. If you choose Close from the local menu, KeyCAD will complete the entity by drawing a line or curve from the entity's first point to its last point.*

Changing Focus

When you are drawing and editing, you will need to move back and forth between *KeyCAD*'s various controls, sometimes in the middle of an operation. You may start an entity by defining a point with the mouse, then continue by defining a point in the Status Bar, then set values in the Edit Bar, then return to the drawing area to complete the operation. You may even want to set up a construction line, zoom into an area of the screen, change the coordinate system, then return to the drawing area to continue an operation already in progress.

Changing Focus with the Mouse

Keep in mind that only one control on the screen can have the focus at a given time. When you move from control to control, you change the focus from one control to another. Generally speaking, users change the focus from one control to another by clicking the control they want to change to. If you are in the drawing area, you can move to the Edit Bar by simply clicking in one of its fields, or move to the Status Bar by clicking in one of its fields. Then you return to the drawing area by clicking there.

The tricky part is that when you return to the drawing area, you will probably want to reactivate it first, so that you can get appropriate feedback while you move your mouse, and only then continue the current operation. For this reason, you need to click twice. The first click reactivates the drawing area, restoring all feedback, and the second click actually continues the current drawing operation.

Changing Focus with the Keyboard

You can also move the focus between controls using the keyboard. Remember these keystrokes:

- If you are in the drawing area or the coordinate fields, you can move the focus to the Edit Bar by pressing <Ctrl>+<E>.
- If you are in the Edit Bar or the coordinate fields, you can move to the drawing area by pressing <Ctrl>+<D>.
- If you are in the Edit Bar or the drawing area, you can return the focus to the coordinate fields by pressing <Ctrl>+<R>.

There are also a couple of alternative ways to get out of the drawing area. You can also just start typing values that you want to go into the Edit Bar: This will change the focus to the Edit Bar, and values you type will be entered in the Edit Bar's first field. This last approach can be very convenient, but it won't work if you are using the Text tool.

NOTE: *If you type values that are not appropriate for a particular Edit Bar field, the field may not accept the values, in which case it will warn you by beeping.*

SEE ALSO: *Defining Points and Specifying Measurements, Finishing and Canceling*

Using the Local Menu

The local menu is context-sensitive: The specific commands and options available in the local menu change, depending on the currently active drawing or editing tool. Many of these commands and options are quite powerful, so it really pays to right-click in the drawing area and see what options are there when you are learning to use a drawing tool.

For example, when you are using the Text tool, the local menu lets you size and rotate the text "on the fly," and specify with great precision exactly where the text will be placed relative to the insertion point.

Command Reference

The Insert menu provides access to tools that let you draw entities and commands that enable you to insert pictures and OLE objects into your drawing.

The following table summarizes the contents of the Insert menu:

Tool	Function
Construction	Create temporary construction lines and circles to use as points of reference.
Point	Draw single-point marker entities in various shapes.
Line	Draw lines, rectangles, polygons, and lines tangent to circles and arcs.
Double Line	Draw double line segments, rectangles, and polygons.
Arc	Draw circular and elliptical arcs.
Circle/Ellipse	Draw circles and ellipses.
Curve	Draw Bezier curves, spline curves, and freehand sketches.
Text	Add single- or multiple-line text to the drawing.
Dimension	Draw dimension entities.
File	Insert a CAD file into an open drawing.
Picture	Insert images (bitmaps) into the drawing.
Object	Insert OLE objects into the drawing.

Construction Tools

Menu: Insert|Construction

The construction tools enable you to place temporary construction lines and circles on your drawing. Constructions are entities used for reference: You can define points in relation to constructions and use them to segment your drawing space.

NOTES: *Constructions exist on their own layer of the drawing, called "$CONSTRUCTION". You can clear this layer with the Clear|Constructions command. You can clear an individual construction with the Edit|Clear|Construction tool.*

The properties of constructions cannot be accessed or changed.

The following table summarizes the Construction tools:

Tool	Function
Horizontal Construction Line	Draw a horizontal construction line.
Vertical Construction Line	Draw a vertical construction line.

Horizontal Construction Line

Menu: Insert|Construction|Horizontal Line

To place a horizontal construction line in your drawing:

1. Activate the Horizontal Construction Line tool.
2. Define a point that you want the line to intersect. *KeyCAD* draws a horizontal construction line that intersects the point you have defined.

Vertical Construction Line

Menu: Insert|Construction|Vertical Line

To place a vertical construction line in your drawing:

1. Activate the Vertical Construction Line tool.
2. Define a point that you want the line to intersect. *KeyCAD* draws a vertical construction line that intersects the point you have defined.

Point

Menu: Insert|Point   **Toolbars**: Point, Insert

You can create five different types of marker points: dot, star, square, cross, or circle. Each type of point is created in the same way.

Dot, Star, Square, Cross, and Circle

Menu: Insert|Point|Dot, Star, Square, Cross, or Circle
Toolbar: Point, Insert

KeyCAD has five styles of points, four of which are essentially similar, except for their shape. The Dot point is different: It stays the same size no matter how close you zoom in or how far you zoom out. This is useful if you want to use it as a point of reference while working in different views of the drawing.

To place a point:

1. Activate the Point tools and select the type of point you want to create.

2. Define a point.

NOTE: *Points cannot be scaled or rotated.*

TIP: *If you create a point of a particular shape and you subsequently want to change the shape, activate the Select tool, double-clicking on the point, and then change its Point Type in the Point property sheet.*

Line

Menu: Insert|Line **Toolbar**: Line, Insert

The Line tools enable you to draw a single line segment, multiple connected segments (multilines), polygons, rectangles, perpendicular lines, parallel lines, and lines tangent to arcs.

The following table summarizes the line tools:

Tool	Function
Single Line	Draw a single line segment.
Multiline	Draw a series of connected line segments.
Polygon	Draw a regular polygon with an arbitrary number of sides.
Irregular Polygon	Draw a polygon with arbitrary segment lengths and angles.
Rectangle	Draw an orthogonal rectangle.
Rotated Rectangle	Draw a rectangle oriented at any angle.
Perpendicular Line	Draw a line perpendicular to an existing line from any point.
Parallel	Draw a line parallel to an existing line, at a specific distance from the line.

Tan Arc Pt	Draw a line tangent to an arc, touching the arc at the line's midpoint.
Tan to Arc	Draw a line tangent to an arc, with the line's second endpoint touching the arc.
Tan from Arc	Draw a line tangent to an arc, with the line's first endpoint touching the arc.
Line 2 Arcs	Draw a line tangent to two arcs, with each endpoint of the line touching one of the arcs.

Single Line

Menu: Insert|Line|Single   **Toolbars**: Line, Insert

Use the Single Line tool to draw a line segment between two defined points:

1. Activate the Single Line tool.

2. Define the line segment's starting point.

3. Do one of the following:

    • Define the line segment's ending point.

    • Set the length and angle of the line segment in the Edit Bar, then press <Enter>.

Multiline

Menu: Insert|Line|Multiline   **Toolbars**: Line, Insert

The Multiline tool enables you to draw a series of connected line segments that comprise one entity. You can create a multiline by defining a series of points. Alternatively, you can set a starting point, then define each line segment in sequence by entering its length and angle in the Edit Bar, pressing <Enter>
to complete each segment of the multiline, and pressing <Ctrl>+<F> to finish
the multiline:

1. Activate the Multiline tool.

2. Define the first point of the multiline entity.

3. Do one of the following:

    • If necessary, reset the snap mode, then define a second point:
      KeyCAD draws a line segment between the two points.

    • Specify the length and angle of the segment in the Edit Bar and
      press <Enter>.

4. Repeat step 3, adding connected segments to the multiline, until
    the multiline is finished.

5. To define the last point in the series, double-click, choose Finish
    from the local menu, or press <Ctrl>+<F> if you are working in
    the Edit Bar.

TIPS: *Instead of choosing Finish, you can have KeyCAD close the
multiline for you by choosing Close from the local menu. This draws a
line segment from the last point of the multiline to the first point.*

*To undo the most recently drawn segment of the multiline, open the local
menu and choose One Step Back.*

NOTE: *Once a multiline is closed, KeyCAD treats it as a polygon, which
means that it will be filled with the color and pattern of the current brush.
You can control brush formatting through the Format|Properties dialog.*

Multiline entity
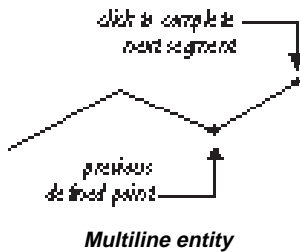
Polygon

Menu: Insert|Line|Polygon   **Toolbars**: Line, Insert

Use the Polygon tool to draw a regular polygon by defining points for the
polygon's center and for one of its vertices:

1. Activate the Polygon tool.

2. Define a point for the center of the polygon.

3. Enter the number of sides for the new polygon in the Sides field on the

Edit Bar.

4. Do one of the following:

   - Define a second point for one of the polygon's vertices.

   - Type values for the polygon's radius and degree of rotation in the Edit Bar. The radius is the distance from the centerpoint to one of the vertices, and the degree of rotation is the angle of a line from the centerpoint to one vertex.
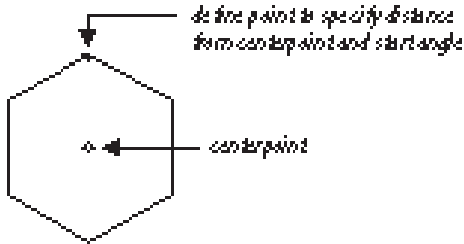
Regular polygon

Irregular Polygon

Menu: Insert|Line|Irregular Polygon   **Toolbars**: Line, Insert

Use the Irregular Polygon tool to draw a polygon with irregular sides and angles.

1. Activate the Irregular Polygon tool.

2. Define a point for the first vertex of the irregular polygon.

3. Define a second point: *KeyCAD* draws a line segment between the two points.

4. Define a third point. *KeyCAD* draws a closed polygon, connecting the three points.

5. Repeat step 4, adding sides and vertices to the irregular polygon.

6. After you have defined the final point, right-click in the drawing area and choose Finish from the local menu.

Alternatively, you can enter the length and angle of each new line segment from the current vertex to the new vertex in the Edit Bar. Press <Enter> to define each new line segment. Press <Ctrl>+<F> to finish the irregular polygon.
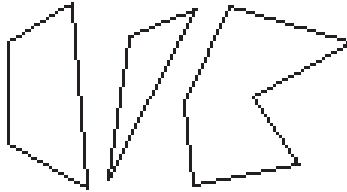
Irregular polygon

Rectangle

Menu: Insert|Line|Rectangle   **Toolbars**: Line, Insert

The Rectangle tool enables you to create a rectangle by defining two diagonally opposite corners. Rectangles created in this way are oriented orthogonally, although they can subsequently be rotated using the editing tools:

1.   Activate the Rectangle tool.

2.   Define a point for one corner of the rectangle.

3.   Do one of the following:

   • Define a point for the diagonally opposite corner.

   • Specify the A (horizontal) and B (vertical) size of the rectangle in the Edit Bar, then press <Enter>.

TIP: *If you use the Edit Bar method, the A size will be drawn upward from the starting point, and the B size will be drawn to the right of the starting point. To draw downward or to the left, use negative values.*

Rotated Rectangle

Menu: Insert|Line|Rotated Rectangle   **Toolbars**: Line, Insert

Use the Rotated Rectangle tool to create a rectangle oriented at any angle. You create rotated rectangles by defining three points:

1.   Define two points to specify the endpoints of a line that forms the rectangle's base.

2.   Do one of the following:

   • Define a third point to specify the distance of the opposite side

of the rectangle from the baseline.

• Type the rectangle's dimensions and angle of rotation into the Edit Bar fields, then press <Enter>.

Rotated rectangle
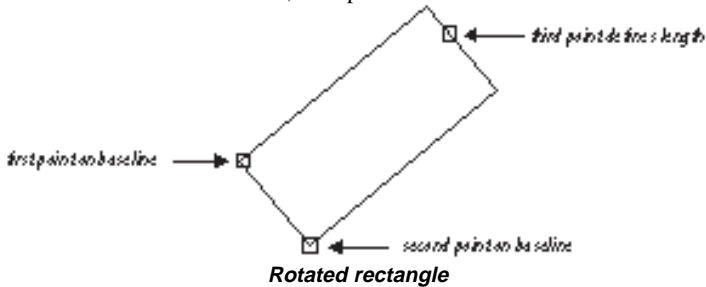
Perpendicular Line

Menu: Insert|Line|Perpendicular Line   **Toolbars**: Line, Insert

Use the Perpendicular Line tool when you want to draw a line perpendicular to an existing line. The new line extends from a defined point to the existing line:

1.   Activate the Perpendicular Line tool.

2.   Click on an existing line segment to which you want to draw the perpendicular line.

3.   Define the point from which you want the perpendicular line to originate.

If the point you define is beyond the end of the existing line segment, *KeyCAD* by default will draw the perpendicular line from this point to the point where the two lines would meet if the existing line segment were extended. To constrain the perpendicular line so that it must meet the existing line segment, right-click in the drawing area and choose Limit to Segment in the local menu.

Parallel Line

Menu: Insert|Line|Parallel   **Toolbars**: Line, Insert

Use the Parallel Line tool to draw a line parallel to an existing line.

1. Activate the Parallel Line tool.

2. Click on or near an existing line. Let's call this the *source line*. As you move the mouse cursor away from the source line, *KeyCAD* draws a line parallel to it.

3. Define a second point, which will be the second endpoint of the new line.

You can also select the source line, then specify the length of the parallel line and its distance from the source line in the Edit Bar. Press <Enter> or <Ctrl>+<F> to finish the parallel line.

TIP: *To create a parallel line that is constrained to be the same length as the source line, right-click in the drawing area with the Parallel Line tool active, then switch on the Keep Length option in the local menu.*

Tan Arc Pt

Menu: Insert|Line|Tan Arc Pt   **Toolbars**: Line, Insert

The Tan Arc Pt tool enables you to draw a line tangent to an arc or circle with the midpoint of the line centered at the point of tangency:

1. Activate the Tan Arc Pt tool.

2. Click on the arc or circle on which you want to draw the line. A line appears tangent to the arc at a point on the arc near the mouse click.

3. Move the mouse to adjust the length of the line. The line length changes in both directions so that the midpoint of the line remains at the point where the line and arc meet. Click to set the line length.

4. A dotted line appears, from the center of the arc or circle to the mouse position, perpendicular to the tangent line. Move the mouse to change the angle of the dotted line and the position where the tangent line touches the arc, then click to set the angle of the tangent line.

Alternatively, you can click on the circle or arc, then specify the length and angle of the tangent line in the Edit Bar, then press <Enter>.

Using the Tan Arc Pt tool

Tan to Arc

Menu: Insert|Line|Tan to Arc   **Toolbars**: Line, Insert

Use the Tan To Arc tool to draw a line tangent to an arc or circle from a defined point.

1.   Activate the Tan to Arc tool.

2.   Define the point from which you want to draw the tangent line.

3.   Click on the side of the arc or circle to which you want to draw the tangent line.

TIP: *To create a tangent line with a fixed length, define the first point of the line, then enter a length in the Edit Bar and check the lockdown box. Return to the drawing area and click on the circle or arc near the point of tangency. If the length you define is too long or too short this won't work—but you can return to the Edit Bar to adjust the length.*

Tan from Arc

Menu: Insert|Line|Tan from Arc   **Toolbars**: Line, Insert

Use the Tan from Arc tool to draw a line tangent from an arc or circle.

1.   Activate the Tan from Arc tool.

2.   Click on the side of the arc or circle from which you want to draw the tangent line.

3.   Do one of the following:

   • Define the point to which you want to draw the tangent line.

   • Specify the length of the line in the Edit Bar, then press <Enter>.

Line 2 Arcs

Menu: Insert|Line|Line 2 Arcs   **Toolbars**: Line, Insert

Use the Line 2 Arcs tool to draw a line tangent to two arcs or circles:

1. Activate the Draw|Line|Line 2 Arcs tool.

2. Click on the side of the first arc or circle from which you want the line drawn tangent.

3. Click on a side of the second arc or circle.

NOTE: *The current snap mode is irrelevant when you use this tool.*
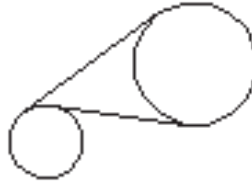
Lines tangent to two circles

Double Line Tools

Menu: Insert|Double Line   **Toolbars**: Double Line, Insert

Use the double line tools to draw two parallel lines as a single entity. The Double Line tools are essentially similar to the Line tools, except that there are no Double Line tools for drawing lines tangent to arcs. The following table summarizes the double line tools:

Tool	Function
Double Line Segment	Draw a double line segment by defining two endpoints.
Double Line Multiline	Draw a series of connected double lines.
Double Line Polygon	Draw a regular double line polygon with an arbitrary number of sides.
Double Line Irregular Polygon	Draw a double line polygon with arbitrary line lengths and angles.
Double Line Rectangle	Draw a horizontal double line rectangle.

Perpendicular Double Line   Draw a double line perpendicular to an existing line.

Double Line Properties

Double Lines have unique properties that you can set in the Double Line property sheet of the Edit|Properties dialog. You can access the default properties for double lines by right-clicking on any double line tool. You can access the properties of a specific double line entity by activating the Select tool and then double-clicking on the entity.

In the Double Line property sheet you can:

- Set the distance between the two lines of the double line.
- Determine whether the endpoints of the double line are open or closed. Each endpoint can be set individually.
- Set the alignment of the double line. Alignment sets whether the nodes (reference points) of the double line are aligned along its "left" line, its "right" line, or along its center.

The alignment of the nodes determines how the double line aligns itself to the coordinate system, and to the grid and other objects when you use snap modes to position the double line.

TIP: *To determine which is the left and which is the right side of a double line, imagine yourself standing on the starting point of the line, facing the endpoint. No matter which way you rotate the line, the right side remains on your right. This is similar to the terms "port" and "starboard" which define directions relative to the fore and aft of a ship. The critical thing is to know the direction in which the double line is drawn.*

Double Line Separation

The separation of a double line (the distance between the two single lines that comprise the double line) is set only in its property sheet. If you stretch or scale a double line, its separation always remains the same. If you want the separation of the double line to change as you resize a selection that contains the double line, first select the double line and choose Explode from the Format menu. The first time you explode a double line, it will become a group of single line segments. (If the double line entity is itself contained within a group, you will need to explode it more than once.) Once the double line is exploded into two single lines, the distance between the two single lines will be scaled along with the selection that contains them.

SEE ALSO: *Double Line Property Sheet*

Double Line Segment

Menu: Insert|Double Line|Single   **Toolbars**: Double Line, Insert

Use the Double Line Segment tool to draw a double line segment between two defined points.

1. Activate the Double Line Segment tool.

2. Define the double line segment's starting point.

3. Do one of the following:

    • Define the double line segment's ending point.

    • Specify the length and angle of the double line segment in the Edit Bar, then press <Enter>.

Double Line Multiline

Menu: Insert|Double Line|Multiline   **Toolbars**: Double Line, Insert

The Double Line Multiline tool enables you to draw a series of connected double line segments. You can create a multiline by defining a series of points. Alternatively, you can set a starting point, then define each double line segment in sequence by entering the length and angle of each segment in the Edit Bar, pressing <Enter> to complete each segment of the multiline.

1. Activate the Double Line Multiline tool.

2. Define the first point of the multiline entity.

3. Do one of the following:

    • If necessary, reset the snap mode, then define a second point: *KeyCAD* draws a double line segment between the two points.

    • Specify the length and angle of the double line segment in the Edit Bar and press <Enter>.

4. Repeat step 3, adding connected double line segments to the multiline, until the multiline is finished.

5. To define the last point in the series, right-click in the drawing area and choose Finish from the local menu, or press <Ctrl>+<F> if you are working in the Edit Bar.

TIPS: *You can have KeyCAD close the multiline for you by choosing Close from the local menu. This draws a double line segment from the last point of the multiline to the first point, resulting in a double line polygon.*

If you want to cancel the previous line segment, open the local menu and choose One Step Back.

Double Line Polygon

Menu: Insert|Double Line|Polygon   **Toolbars**: Double Line, Insert

Use the Double Line Polygon tool to draw a regular double line polygon by defining points for the polygon's center and for one of its vertices (corners).

1.   Activate the Double Line Polygon tool.

2.   Enter the number of sides for the double line polygon in the Sides field of the Edit Bar.

3.   Define the centerpoint of the polygon.

4.   Do one of the following:

   • Define a second point for one of the polygon's vertices.

   • Type values for the polygon's radius and degree of rotation in the Edit Bar. The radius is the distance from the centerpoint to one of the vertices, and the degree of rotation is the angle of a line from the centerpoint to one vertex.

Double Line Irregular Polygon

Menu: Insert|Double Line|Irregular Polygon   **Toolbars**: Double Line, Insert

Use the Double Line Irregular Polygon tool to draw a double line polygon with irregular line segment lengths and angles:

1.   Activate the Double Line Irregular Polygon tool.

2.   Define the first point of the irregular polygon.

3.   Define a second point: *KeyCAD* draws a double line between the two points.

4. Define a third point. *KeyCAD* draws a closed polygon, connecting the three points.

5. Repeat step 4, adding sides and vertices to the irregular polygon.

6. To define the last point in the series, choose Finish from the local menu.

Alternatively, you can enter the length and angle of each line segment from the current point in the Edit Bar. Press <Enter> to define each new line. Press <Ctrl>+<F> to finish the irregular polygon.

Double Line Rectangle

Menu: Insert|Double Line|Rectangle   **Toolbars**: Double Line, Insert

This tool enables you to create a double line rectangle by defining two diagonally opposite corners. Rectangles created in this way are orthogonal, although they can subsequently be rotated using editing tools.

1. Activate the Double Line Rectangle tool.

2. Define the first point.

3. Do one of the following:

   • Define a point for the diagonally opposite corner.

   • Specify the vertical and horizontal dimensions of the rectangle in the Edit Bar, and then press <Enter>.

NOTE: *If you use the Edit Bar method, the A size will be drawn upward from the starting point, and the B size will be drawn to the right of the starting point. You can draw downward and to the left by entering negative values.*

Perpendicular Double Line

Menu: Insert|Double Line|Perpendicular Line   **Toolbars**: Double Line, Insert

Use the Perpendicular Double Line tool when you want to draw a double line perpendicular to an existing line. The double line extends from a defined point to the existing line.

1. Activate the Perpendicular Line tool.

2. Click on an existing line segment to which you want to draw the perpendicular double line. (This can be one of the sides of another double line.)

3. Define the point from which you want the perpendicular line to originate.

Tip: If the point you define is beyond the end of the existing line segment, KeyCAD will normally draw the double perpendicular line from this point to where it would meet the existing line segment if the existing line segment were extended. If you want to constrain the perpendicular line so that it will not be drawn beyond the ends of the existing line segment, right-click in the drawing area to open the local menu, then choose Limit to Segment.

Circle/Ellipse

Menu: Insert|Circle/Ellipse   **Toolbar**: Ellipse, Insert

The Circle/Ellipse menu item opens a submenu containing tools for drawing circles and ellipses. You can also access these tools from the Ellipse toolbar, and on the Ellipse flyout of the Insert toolbar.

The table on the following page summarizes the Circle/Ellipse tools:

Tool	Function
Center and Point	Draw a circle by specifying its centerpoint and radius.
Concentric	Draw a set of circles that share the same centerpoint.
Double Point	Draw a circle by defining two opposite points on its perimeter.
Circle Tan to Arc	Draw a circle tangent to another circle or arc.
Circle Tan to Line	Draw a circle tangent to a line.
Ellipse	Draw an ellipse by defining its bounding rectangle.
Rotated Ellipse	Draw an ellipse rotated at any angle.
Ellipse Fixed Ratio	Draw an ellipse, specifying the ratio of the major axis length to the minor axis length.

Center and Point

Menu: Insert|Circle/Ellipse|Center and Point   **Toolbar**: Ellipse, Insert

Use the Center and Point tool to create a circle by defining its center and a point on its perimeter.

1. Activate the Center and Point tool.

2. Define a point for the center of the circle.

3. Do one of the following:

   • Define a point on the perimeter of the circle.

   • Enter the radius of the circle in the Radius field, or the circumference of the circle in the Circumf. field on the Edit Bar, then press <Enter>.

Concentric

Menu: Insert|Circle/Ellipse|Concentric   **Toolbar**: Ellipse, Insert

Use the Concentric tool to draw concentric circles, that is, circles that share the same centerpoint. You can create as many concentric circles as you need at one time:

1. Activate the Concentric tool.

2. Define a centerpoint for the circles.

3. Do one of the following:

   • Define a point on the perimeter of the circle.

   • Enter the circle's radius in the Radius field on the Edit Bar and press <Enter>.

   KeyCAD will draw a circle with the specified center and radius.

4. Repeat step 3, drawing as many concentric circles as required.

5. Finish the concentric circles by choosing Finish from the local menu, or by pressing <Ctrl>+<F> if you are working in the Edit Bar.

You can also enter the radius or circumference of each concentric circle in succession in the Edit Bar. Press <Enter> to complete individual circles. Press <Ctrl>+<F> to complete the set of concentric circles.

TIP: *To draw a circle concentric to an existing circle, choose the Center and Point tool, then use Snap to Arc Center to snap to the center of the existing circle.*

Double Point

Menu: Insert|Circle/Ellipse|Double Point   **Toolbar**: Ellipse, Insert

The Double Point tool lets you draw a circle by defining two opposite points on the circle's perimeter:

1. Activate the Double Point tool.

2. Define a point that will be on the perimeter of the circle.

3. Do one of the following:

    • Define a second point, 180 degrees from the first point on the perimeter of the circle.

    • Specify the diameter (or circumference) of the circle and the angle of the line in the Edit Bar, then press <Enter>.

Circle Tan to Arc

Menu: Insert|Circle/Ellipse|Tan to Arc   **Toolbar**: Ellipse, Insert

Use the Circle Tan to Arc tool to draw a circle tangent to an arc or to another circle.

1. Activate the Circle Tan to Arc tool.

2. Click on the existing arc or circle to which you want the circle to be tangent.

3. Do one of the following:

    • Define the centerpoint of the circle.

    • Enter the radius of circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click once more to continue to the next step.

4. Move the mouse along the perimeter of the arc to position the circle on the arc. If you move the mouse along the inside of the arc, the circle will move to the inside of the arc; if you move the

mouse along the outside of the arc, the circle will move to the outside. Once the circle is in place, click the mouse to fix its position.

NOTE: *After clicking on the circle or arc (step 2), you can set the radius or circumference of the circle you are drawing in the Edit Bar and press <Enter>, finishing the entity. Be aware that if you use this method, you will not be able to reposition the circle on the arc using this tool, so make sure that you place it where you want it with the first click.*
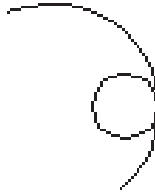
Circle tangent to arc

Circle Tan to Line

Menu: Insert|Circle/Ellipse|Tan to Line   **Toolbar**:
    Ellipse, Insert

Use the Circle Tan to Line tool to draw a circle tangent to an existing line.

1. Activate the Circle Tan to Line tool.

2. Click on the line to which you want the circle to be tangent.

3. Do one of the following:

    • Define the centerpoint of the circle.

    • Enter the radius of circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click once more to continue to the next step.

4. Move the mouse along the line to position the circle, then click (using any snap mode) to fix the circle's position.

NOTE: *After clicking on the line (step 2), you can set the radius or circumference of the circle you are drawing in the Edit Bar and press <Enter>, finishing the entity. Be aware that if you use this method, you will not be able to reposition the circle on the line using this tool, so make sure that you place it where you want it with the first click.*
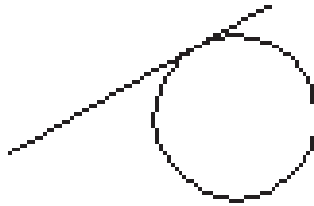
Circle tangent to line

Ellipse

Menu: Insert|Circle/Ellipse|Ellipse   **Toolbar**: Ellipse, Insert

The Ellipse tool lets you create an ellipse by defining a bounding rectangle for the ellipse. The axes of the ellipse will be orthogonal; however, you can rotate the ellipse after you have created it using the editing tools.

1. Activate the Ellipse tool.

2. Define the first point or corner of the bounding rectangle.

3. Do one of the following:

    • Define the diagonally opposed corner of the bounding rectangle.

    • Specify the lengths of the ellipse's major and minor axes in the Edit Bar, then press <Enter>.

Rotated Ellipse

Menu: Insert|Circle/Ellipse|Rotated Ellipse   **Toolbar**: Ellipse, Insert

The Rotated Ellipse tool enables you to create an ellipse tilted at any angle.

1. Activate the Rotated Ellipse tool.

2. Define the centerpoint of the ellipse.

3. Define a point to specify an angle and distance from the centerpoint. This creates an axis from the defined point, through the centerpoint, to an opposite point the same distance from the centerpoint. You can view this axis on the screen as you adjust the position of the point using the mouse or Coordinate Fields.

4. Define a second point to specify the length of the opposite axis.

This axis will of course be perpendicular to the first axis.

Alternatively, you can define the centerpoint of the ellipse, specify the length and angle of the major axis and the length of the minor axis in the Edit Bar, then press <Enter>.

Rotated ellipse
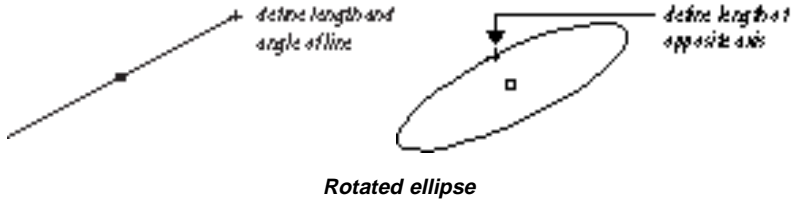
Ellipse Fixed Ratio

Menu: Insert|Circle/Ellipse|Ellipse Fixed Ratio   **Toolbar**: Ellipse, Insert

The Ellipse Fixed Ratio tool lets you create an ellipse by specifying its aspect ratio; that is, the ratio of the length of its major axis to the length of its minor axis. For example, if the ratio is 2 then the major axis will be twice as long as the minor axis (the ellipse will be twice as long as it is wide). The axes of the ellipse will initially be horizontal and vertical; however, you can subsequently rotate the ellipse using editing tools.

1. Activate the Ellipse Fixed Ratio tool.

2. Define the centerpoint of the ellipse.

3. Enter the aspect ratio in the a:b field on the Edit Bar, then lock down this field. To return to the drawing area, click there or press <Ctrl>+<D>.

4. Define any point of the perimeter of the ellipse.

Axes of a fixed-ratio ellipse
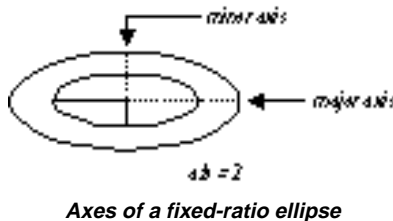
Arc Tools

Menu: Insert|Arc   **Toolbars**: Arc, Insert

Drawing arcs is very similar to drawing circles. Generally, the arc tools work precisely the same way as the circle tools, except that they add one extra step: defining the start and end angles of the arc.

NOTE: *Arcs are always drawn in a counterclockwise direction.*
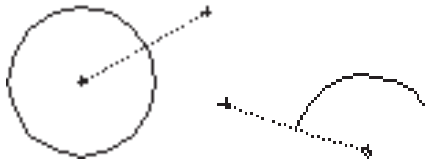
Defining the starting and ending points of an arc

The following table summarizes the arc drawing tools:

Tool	Function
Center and Point Arc	Draw a circular arc by specifying its centerpoint and a point on its circumference.
Concentric Arc	Draw a set of concentric circular arcs that share the same centerpoint.
Double Point Arc	Draw a circular arc by defining the endpoints of its diameter.
Arc Tan to Arc	Draw a circular arc tangent to a circle or arc.
Arc Tan to Line	Draw a circular arc tangent to a line.
Triple Point 1-2-3	Draw a circular arc by defining (1) its starting point, (2) a point on its perimeter, and (3) its ending point.
Elliptical Arc	Draw an elliptical arc by defining its bounding rectangle.
Rotated Elliptical Arc	Draw an elliptical arc rotated at any angle.
Elliptical Fixed Ratio Arc	Draw an elliptical arc by setting its aspect ratio and defining either axis.

Circumference and Length of Arcs

When you are drawing arcs using the Edit Bar, you will need to distinguish between the Circumf. field and the Length field.

The Circumf. field holds that value that represents the circumference of the arc circle, that is, the length of the circle that the arc would form if it were closed. This field and the Radius field are mutually dependent: Changing one changes the other. The Length field is the actual length of the arc, taking into account its starting and ending angles.

Combining Edit Bar and Mouse When Drawing Arcs

Arcs are relatively complex entities, and it's likely that you'll want to use various combinations of the Edit Bar and the mouse to create them. This manual won't be able to cover all the different possible combinations in the following sections; it will cover just the most likely combinations.

TIP: *Keep in mind that if you specify values in the Edit Bar, you must lock the fields down before returning to the drawing area to finish the entity. If you don't, the values that you typed in the Edit Bar will change when you re-enter the drawing area.*

Center and Point Arc

Menu: Insert|Arc|Center and Point   **Toolbars**: Arc, Insert

Use the Center and Point tool to create an arc by defining its center, a point on its circumference, and its starting and ending angles.

1. Activate the Center and Point tool.

2. Define the centerpoint of the arc.

3. Define a point at a distance from the centerpoint to specify the radius of the arc. *KeyCAD* draws a circle with the specified radius and centerpoint. A dotted line appears, extending from the centerpoint of the circle to the cursor.

4. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define a point for the center of the circle, enter the radius (or circumference) of the arc and its starting and ending angles (or length) in the Edit Bar, and then press <Enter>.

You can also combine the two approaches. For example, you could define the center and radius of the arc using the mouse, then specify the starting and ending angles in the Edit Bar.

Concentric Arc

Menu: Insert|Arc|Concentric   **Toolbars**: Arc, Insert

Use the Concentric tool to draw concentric arcs; that is, arcs that share the same centerpoint. These concentric circles are a series of center and point arcs, created in succession. You can create as many concentric arcs as necessary at one time.

1. Activate the Concentric Arc tool.

2. Define a centerpoint for the arcs.

3. Define a point on the circumference of the arc. *KeyCAD* will draw a circle with the specified center and radius. A dotted line will appear, extending from the centerpoint of the circle to the cursor.

4. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

5. Repeat steps 3 and 4, drawing as many concentric arcs as required.

6. Once you have all the concentric arcs you need, complete the set of concentric arcs by choosing Finish from the local menu, double-clicking, or pressing <Enter>.

You can also enter the radius (or circumference) and the starting and ending angles (or length) of each concentric arc in succession in the Edit Bar. Press <Enter> to complete individual circles. Press <Ctrl>+<F> to complete the set of concentric circles.

Concentric arcs

Double Point Arc

Menu: Insert|Arc|Double Point   **Toolbars**: Arc, Insert

The Double Point tool lets you draw an arc by defining the endpoints of its diameter.

1. Activate the Double Point tool.

2. Define the starting point of a line that will pass through the center of the arc.

3. Define the endpoint of this line. *KeyCAD* draws a circle that passes through the two defined points. A dotted line appears, extending from the centerpoint of the circle to the cursor.

4. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define the first point of the arc's diameter, then specify all the quantities that define the arc in the Edit Bar.

You can also combine the two approaches. For example, you could define the center and radius of the arc using the mouse, then specify the starting and ending angles in the Edit Bar.

Arc Tan to Arc
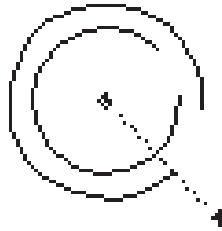
Menu: Insert|Arc|Tan to Arc   **Toolbars**: Arc, Insert

Use the Arc Tan to Arc tool to draw an arc tangent to an existing arc or circle.

1. Activate the Arc Tan to Arc tool.

2. Click on the existing arc or circle to which you want the arc to be tangent.

3. Do one of the following:

    • Define the centerpoint of the arc you are drawing.

    • Enter the radius of circumference of the circle in the Edit Bar, then *lock down* the field in which you typed the value, then return to the drawing area by clicking there (or press <Ctrl>+<D>). Click to continue to the next step.

4. Move the mouse along the perimeter of the arc to position the circle on the arc. If you move the mouse along the inside of the arc, the circle will move to the inside of the arc; if you move the mouse along the outside of the arc, the circle will move to the outside. Once the circle is in place, click the mouse to fix its position. A dotted line appears, extending from the centerpoint of the circle to the cursor.

5. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.

6. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

NOTE: *After clicking on the circle or arc (step 2), you can set the radius (or circumference) of the arc, as well as its starting and ending angles (or length) in the Edit Bar, then press <Enter> to finish the entity. Be aware that if you use this method, you will not be able to reposition the circle on the arc using this tool, so make sure that you place it where you want it with the first click.*

Arcs tangent to a circle

Arc Tan to Line

Menu: Insert|Arc|Tan to Line   **Toolbars**: Arc, Insert

Use the Arc Tan to Line tool to draw an arc tangent to an existing line.

1. Activate the Arc Tan to Line tool.

2. Click on the line to which you want the arc to be tangent.

3. Do one of the following:

    • Define the centerpoint of the arc you are drawing.

    • Enter the radius of circumference of the circle in the Edit Bar,
      then *lock down* the field in which you typed the value, then
      return to the drawing area by clicking there (or press
      <Ctrl>+<D>). Click to continue to the next step.

4. Move the mouse to either side of the line and define a point to
    position the arc on the line. A dotted line appears, extending
    from the centerpoint of the circle to the cursor.

5. Move the cursor to change the dotted line to the angle at which
    you want the arc to start, then click to set the arc's starting angle.

6. Move the cursor counterclockwise to draw the arc, then click the
    mouse button to define the ending angle.

NOTE: *After clicking on the line (step 2), you can set the radius (or
circumference) of the arc, as well as its starting and ending angles (or
length) in the Edit Bar, then press <Enter> to finish the entity. Be aware
that if you use this method, you will not be able to reposition the circle on
the line using this tool, so make sure that you place it where you want it
with the first click.*

Arcs tangent to a line

Triple Point 1-2-3

Menu: Insert|Arc|Triple Point 1-2-3   **Toolbars**: Arc, Insert

The Triple Point 1-2-3 tool lets you draw an arc by defining three points: a starting point, a middle point, and an ending point. The arc will touch all three of the points.

1.   Activate the Triple Point 1-2-3 tool.

2.   Define the starting point of the arc.

3.   Define a point through which the arc will pass.

4.   Define the ending point of the arc.

Triple point arc 1-2-3

Arcs tangent to three circles

Elliptical Arc

Menu: Insert|Arc|Elliptical   **Toolbar**: Arc, Insert

The Elliptical Arc tool lets you create an elliptical arc by defining its bounding rectangle. The lengths of the minor and major axes of the arc are determined by the lengths of the sides of the bounding rectangle. The axes of the arc will be orthogonal; however, you can rotate the arc after you have created it using the editing tools:

1.   Activate the Ellipse tool.

2.   Define a point for the first corner of the bounding rectangle.

3.   Define a point for the diagonally opposite corner of the bounding rectangle. *KeyCAD* draws an ellipse bounded by the rectangle that you have defined. A dotted line appears, extending from the centerpoint of the ellipse to the cursor.

4. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.

5. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

Alternatively, you can define the starting point of the elliptical arc, then specify its major and minor axes and starting and ending angles in the Edit Bar. You can also combine use of the mouse and Edit Bar.

Rotated Elliptical Arc

Menu: Insert|Arc|Rotated Elliptical   **Toolbar**: Arc, Insert

The Rotated Elliptical Arc tool enables you to create an elliptical arc tilted at
any angle:

1. Activate the Rotated Elliptical Arc tool.

2. Define the centerpoint of the arc.

3. Define a point to specify an angle and distance from the centerpoint. This creates an axis from the defined point, through the centerpoint, to an opposite point the same distance from the centerpoint. This is the major axis of the elliptical arc.

4. Define a third point to specify the length of the minor axis (perpendicular to the major axis). *KeyCAD* draws an ellipse with the specified major and minor axes. A dotted line appears, extending from the centerpoint of the ellipse to the cursor.

5. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.

6. Move the cursor counterclockwise to draw the arc, then click the mouse button to define the ending angle.

You can also define the centerpoint of the ellipse, then specify the length and angle of the major axis and the length of the minor axis, as well as the starting and ending angles of the arc, in the Edit Bar.
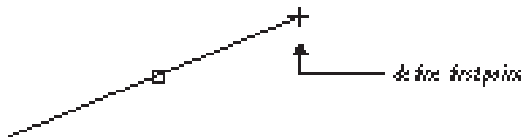
Rotated elliptical arc

Elliptical Fixed Ratio Arc

Menu: Insert|Arc|Elliptical Fixed Ratio   **Toolbar**: Arc, Insert

The Elliptical Fixed Ratio Arc tool lets you create an elliptical arc by specifying its aspect ratio; that is, the ratio of the length of its major axis to the length of its minor axis. For example, if the ratio is 2 the major axis will be twice as long as the minor axis. The axes of the elliptical arc will initially be orthogonal; however, you can subsequently rotate the elliptical arc using the editing tools.

1. Activate the Elliptical Fixed Ratio Arc tool.
2. Define the centerpoint of the arc.
3. Type the aspect ratio of the arc into the Ratio field of the Edit Bar.
4. Define any point of the perimeter of the ellipse. *KeyCAD* draws an ellipse with the specified major and minor axes. A dotted line appears, extending from the centerpoint of the ellipse to the cursor.
5. Move the cursor to change the dotted line to the angle at which you want the arc to start, then click to set the arc's starting angle.
6. Move the cursor to draw the arc, then click the mouse button to define the ending angle.

As an alternative to steps 5 and 6, you can specify the start and end angles of the arc in the Edit Bar, then press <Enter>.

Curve Tools

Menu: Insert|Curve   **Toolbar**: Curve, Insert

The following table summarizes the curve tools:

Tool	Function
Bezier	Connects two endpoints; gravitates toward control points along its path.
Spline	Continuous curve that connects a series of points.
Sketch	Freehand multiline drawn with mouse.

Curve Properties

Curve's have special properties that can be set in the Curve property sheet of the Properties dialog. This property sheet lets you show or hide the frame of a Bezier or Spline curve, transform a Bezier curve into a spline curve, or vice versa. You can access the default properties of curves by property sheet by double-clicking on a curve with the Select tool, or by selecting the curve and then choosing Format|Properties.

TIP: *You can close a Bezier or Spline curve by choosing Close from the local menu instead of Finish. This draws a curved line between the two endpoints. You can also close curves or break open closed curves using Node Edit mode, employing the same technique that you use on multilines and polygons.*

SEE ALSO: *Breaking Polygons, Closing Multilines, Curve Property Sheet*

Bezier Curve

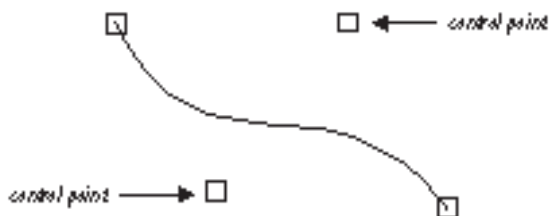**Menu**: Insert|Curve|Bezier   **Toolbar**: Curve, Insert

To create a Bezier curve, define points along the intended path of the curve. The curve will span the two endpoints and be pulled toward the intermediate points, which act as control points along its path.

1. Activate the Bezier Curve tool.
2. Define the first endpoint of the curve.
3. Continue to define points along the path of the curve.

4.   To define the final point, right-click and choose Finish from the local menu. If you want to close the curve, choose Close instead of Finish.

If at any point you wish to undo the segment of the curve that you just drew, right-click in the drawing area and choose One Step Back from the local menu.

An alternative to defining points along the path of the curve is to specify the length and angles of the line segments between the points, which form the frame of the curve. You can do this in the Edit Bar, pressing <Enter> to complete each segment of the frame, then pressing <Ctrl>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.

Bezier curve and control points

Spline Curve

Menu: Insert|Curve|Spline   **Toolbar**: Curve, Insert

To create a Spline curve, define points along the intended path of the curve. The curve will pass through the defined points.

1.   Activate the Spline Curve tool.

2.   Define the first endpoint of the curve.

3.   Continue to define points along the path of the curve.

4.   To define the final point choose Finish from the local menu. If you want to close the curve choose Close instead of Finish.

If at any point you wish to undo the segment of the curve that you just drew, right-click in the drawing area and choose One Step Back from the local menu.

An alternative to defining points along the path of the curve is to specify the length and angles of the line segments between the points, which form the frame of the curve. You can do this in the Edit Bar, pressing <Enter> to complete each segment of the frame, then pressing

<Ctrl>+<F> to complete the entire curve. This procedure is essentially similar to drawing a multiline.
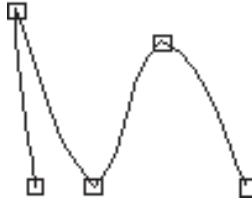
Spline curve passes through points along its path

Sketch Curve

Menu: Insert|Curve|Sketch   **Toolbar**: Curve, Insert

The Sketch tool is used for freehand drawing.

1. Activate the Sketch tool.

2. Click the mouse to define the starting point of the sketch.

3. Move the mouse to draw a freehand polyline.

4. Click the mouse to define the final endpoint of the sketch polyline.

Text

Menu: Insert|Text   **Toolbar**: Insert

The features for creating, formatting, and manipulating text in *KeyCAD* 3.0 are deceptively powerful; however, entering text and placing it in your drawing is simple and intuitive:

1. Activate the Text tool.

2. If you wish to set the font, size, and attributes of text prior to typing, display the Text Format toolbar (using View|Toolbars) and choose settings from the toolbar. As an alternative, you can set text properties by right-clicking on the Text tool, then changing the properties in the Text property sheet. You can use any TrueType font installed in your system.

3. Click in the drawing area at the point where you want to place your text.

4. Type the text that you want to place, using the <Backspace> key to make corrections. Press <Shift>+<Enter> to add a new line of text. To finish the entity, press <Enter>, or right-click and choose Finish from the local menu.

NOTE: *For information about installing TrueType fonts, see your Windows documentation.*

Text Properties

The Text tool has a special set of properties that are accessible through the Text property sheet. You can access the default properties of the Text tool by right-clicking on the tool. You can access the properties of selected text by right-clicking on the selection and then choosing Properties from the local menu. The Text property sheet has options for setting font, style, text height, and color.

TIP: *To edit the contents of the text, select the text object and choose Format/Properties, then edit the text in the Attribute field of the General property sheet. To reformat the text, use the Text property sheet in the same dialog.*

SEE ALSO: *Manipulating Text, Formatting Text, Text Property Sheet*

Text Height and Angle Options

The local menu offers options for changing the height and angle of text "on the fly" while the Text tool is active.

If you right-click and choose Height from the local menu, a rectangle will appear. Move the mouse to position the lower-right corner of the box until it is the desired size. Click the mouse to set the text height, then continue entering text.

If you choose Angle from the local menu, a rectangle will appear. Move the mouse to rotate the rectangle, then click the mouse. You can now continue entering text into the angled text frame.

You can also change the text height and by typing values into the Height and Angle fields of the Edit Bar.

TIP: *To use Ortho while rotating text, hold down the <Shift> key.*

NOTE: *The editing and formatting tools provide other options for rotating and resizing text more precisely after it has been drawn.*

SEE ALSO: *Manipulating Text, Formatting Text, Text Property Sheet*

Text Alignment Options

The text alignment options of the Text tool give you precise control over how the text is aligned relative to the point where it is inserted in your drawing. To access these options, activate the text tool, right-click in the drawing area, and then choose Align (the command will be followed by two capital letters that vary, depending on the current alignment setting).
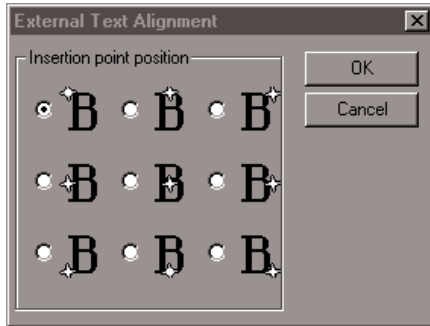
Text Alignment options of Text tool

The figures next to each alignment setting show you graphically how the text is positioned relative to the insertion point, which is represented by a star. There are three vertical positions (Top, Center, and Bottom) and three horizontal positions (Left, Middle, and Right). When you re-open the local menu with the Text tool active, the letters next to the command represent the current settings. TL, for example, means that the insertion point will be at the top-left corner of the text.

Dimension Tools

Menu: Insert|Dimension   **Toolbars**: Dimension, Insert

KeyCAD contains a variety of dimension tools that you can use to display the measurements of lines and angles. Dimensions generally consist of a dimension line or arc, extension lines extending to the object being measured, and dimension text. Leader dimensions consist of dimension text and a simple leader line. *Dimensioning* an entity means to display a dimension that measures or otherwise marks that entity.

By default, dimension text contains a measurement of the dimensioned object or coordinates in current World units. (Leader dimensions, which contain text labels, are the exception to this rule.) You can override the default text of a dimension by specifying the text in the Edit Bar prior to finishing the dimension.

SEE ALSO: *Dimension Property Sheets*

The following table summarizes the dimension tools:

Tool	Function
Horizontal Dimension	Draw a linear dimension showing horizontal distance.
Vertical Dimension	Draw a linear dimension showing vertical distance.
Parallel Dimension	Draw a linear dimension showing absolute distance between two points.
Angular Dimension	Draw a angular dimension showing the angle formed by two lines, by points on the perimeter of a circle or arc, or the angle formed by any two points relative to a defined vertex.
Radius Dimension	Draw a dimension indicating the radius of an arc or circle.
Diameter Dimension	Draw a dimension indicating the diameter of an arc or circle.
Leader Dimension	Draw a simple line leader with label text.

Manual, Segment, and Entity Dimensioning

Horizontal, vertical, and parallel dimensions are collectively known as linear dimensions. When you create a linear dimension, you have three primary choices:

- You can draw the dimension manually, using the Coordinate Fields, Edit Bar, or the mouse and snap modes.
- You can associate the dimension with a line segment.
- You can associate the dimension with an entire entity (such as a multiline or a group).

To choose one of these three modes:

1. Activate the dimension tool that you want to use.
2. Right-click in the drawing area to open the local menu.
3. Do one of the following:

    • Click to place a check mark next to Segment Dimensioning to associate the dimension with a line segment.

    • Click to place a check mark next to Entity Dimensioning to associate the dimension with an entity.

    • If you want to create the dimension manually, uncheck both Segment Dimensioning and Entity Dimensioning.

Once you have selected a dimensioning mode, the mode is saved with the drawing and remains in effect for that drawing unless you explicitly change it.

For further information on drawing linear dimensions, read the following sections on the individual dimension tools.

TIP: *If you generally prefer one of the dimensioning modes over the others, you can save it in a drawing template.*

SEE ALSO: *Horizontal Dimension; Parallel Dimension; Vertical Dimension*

Associative Dimensions

Dimensions can be *associated* to objects. This means that you can freely move, rotate, or resize the objects with which the dimension is associated, and the dimension will retain its position relative to the objects, and the dimension text will change automatically to reflect changes resulting from the objects being resized or moved (provided that you did not override the automatic text in the Edit Bar when you created the dimension).

If you want a dimension to be associated with an object, you first need to be sure that the Associative Dimension option is turned on. To do this, first right-click on any dimension tool to bring up the Properties dialog, which will contain the property sheets for default dimension properties. If there is no check mark in the Associative Dimension check box, then click in the box. Click OK to close the dialog.

Once this box is checked, then any dimension that is "connected" to an object will be associated with the object. You can accomplish this in the following ways:

- Use Segment Dimensioning.
- Use Entity Dimensioning.
- Use any snap mode to snap the dimension to an existing object.
- Use a dimension tool that automatically connects to an entity. If you dimension an angle on a circle, for example, the dimension will be associated with the circle. If you dimension an angle between two lines, the dimension will be associated with both lines.

TIPS: *To tell whether a dimension is associated with an entity, select the entity. Any dimensions associated with that entity will be colored blue.*

If you want to turn off default associative dimensions, first right-click on a dimension tool to bring up the Properties dialog. Then, in the Format property sheet, uncheck the Associative Dimensions check box.

Setting Dimension Properties

The Dimension property sheets provide a large number of options for controlling the format of dimensions. You can set the default properties of dimensions by right-clicking on a dimension tool and then choosing settings in the Dimension property sheets. These settings are saved in your drawing or template and remain in effect until you explicitly change them.

If you want to set the properties of a dimension or a set or dimensions, activate the Select tool, select the dimensions that you want to format, then right-click on the selection and choose Properties from the local menu to bring up the Properties dialog.

SEE ALSO: *Dimension Property Sheets*

Horizontal Dimension

Menu: Insert|Dimension|Horizontal   **Toolbars**: Dimension, Insert

Use the Horizontal Dimension tool to display the horizontal length of an entity, or to display the Horizontal distance between any two points that you define in the drawing space:

1. Activate the Horizontal Dimension tool.

2. If desired, change the current dimensioning mode, as described in the previous section "Manual, Segment, and Entity Dimensioning."

3. Do one of the following:

    • If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.

    • If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)

4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.

Horizontal dimensions

Vertical Dimension

Menu: Insert|Dimension|Vertical   **Toolbars**: Dimension, Insert

Use the Vertical Dimension tool to display the Vertical length of an entity, or to display the vertical distance between any two points that you define in the drawing space:

1. Activate the Vertical Dimension tool.

2. If desired, change the current dimensioning mode, as described in the previous section "Manual, Segment, and Entity Dimensioning."

3. Do one of the following:

    • If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.

    • If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)

4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.
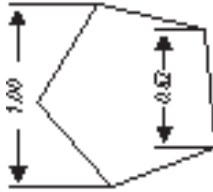
Vertical dimensions

Parallel Dimension

Menu: Insert|Dimension|Parallel   **Toolbars**: Dimension, Insert

Use the Parallel Dimension tool to display the absolute length of an entity (the distance between the two points on the entity that are farthest apart).

1. Activate the Parallel Dimension tool.

2. If desired, change the current dimensioning mode, as described in the previous section "Manual, Segment, and Entity Dimensioning."

3. Do one of the following:

    • If Segment or Entity Dimensioning mode is active, click on the segment or entity that you want to dimension.

    • If you are dimensioning manually, define two points, using any snap mode or the Coordinate Fields. (You can also specify the length and angle of a line connecting the two points in the Edit Bar.)

4. Define a final point to determine the direction and length of the dimension's extension lines. (You can also specify this length directly in the Edit Bar.)

To override the default, automatic dimension text, type the text you want in the Edit Bar prior to finishing the dimension.

NOTE: *When dimensioning a circle or arc, you can position the parallel dimension at any angle.*
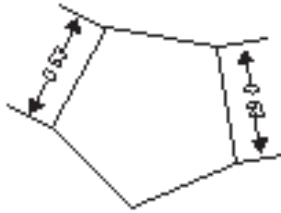
Parallel dimension

Angular Dimension

Menu: Insert|Dimension|Angular   **Toolbars**: Dimension, Insert

Use this tool to dimension angles in your drawing. You can dimension:

- The angle formed by any two lines.
- The angle of an arc.
- The angle formed by any two points on the perimeter of a circle.
- A freeform angle that you create by defining a vertex node and two points.

These techniques are discussed in the following sections.
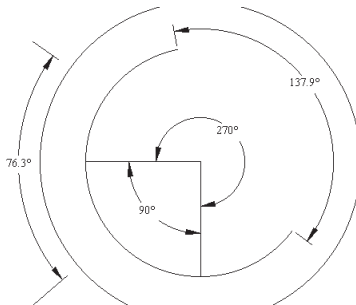
Angular dimensions

Dimensioning an Angle Formed by Two Lines

To dimension the angle formed between two lines:

1.   Activate the Angular Dimension tool.

2. Click on the first line.

3. Click on the second line. If you now move the mouse cursor between the two lines, *KeyCAD* forms a dimension showing the inner angle. If you move the mouse cursor outside the two lines, a dimension will be formed showing the outer angle.

4. Click to specify the position of the dimension arc.

Dimensioning Angles on Circles

To dimension the angle formed between any two points on the perimeter of a circle:

1. Activate the Angular Dimension tool.

2. Click on the circle that you want to dimension. A rubber-band line will appear, extending from the center of the circle to the mouse cursor.

3. Define a second point, which forms an angle with the first point you defined, with the centerpoint of the circle as the angle's vertex.

4. Define a third point to specify the location of the dimension arc. If you specify a location between the first two points, you will dimension the inner angle; if you specify a location outside the two points, you will dimension the outer angle. You can locate the dimension arc inside or outside the circle.

Dimensioning Angles on Arcs

To dimension the angle of an arc:

1. Activate the Angular Dimension tool.

2. Click on the arc that you want to dimension.

3. Click to specify the location of the dimension arc. You can dimension either the inner or outer arc, and you can place the dimension either inside or outside the arc that you are dimensioning.

NOTE: *When you use the Angular Dimension tool on an arc, KeyCAD automatically dimensions the angle formed by the endpoints of the arc. If you want to dimension the angle formed by any other two points on the perimeter of the arc, you can employ the Angle Node technique described in the following section, using snap modes to define points at the center and on the perimeter of the arc.*

Dimensioning Freeform Angles

You can dimension the angle formed by any two points in the drawing space relative to a defined vertex, using the Angle Node option from the local menu:

1. Activate the Angular Dimension tool.

2. Right-click in the drawing area to display the local menu, then choose Angle Node.

3. Define a point, using any snap mode or the Coordinate Fields, for the vertex of the angle.

4. Define two more points to form an angle with the Vertex Node.

5. Define a point to position the dimension arc. You can measure either the inner or outer angle formed by the three points.

Radius Dimension

Menu: Insert|Dimension|Radius   **Toolbars**: Dimension, Insert

Use the Radius Dimension tool to dimension the radius of an arc or circle.

1. Activate the Radius Dimension tool.

2. Click on the arc or circle that you want to dimension. *KeyCAD* draws a dimension from the center of the circle or arc to its perimeter.

3. Define a second point to position the dimension on the perimeter of the circle, and to specify the position of the dimension text. The text can be positioned either inside or outside the circle or arc.

Alternatively, you can click on the circle or arc that you want to dimension, specify the text, angle, and length of the dimension in the Edit Bar, then press <Enter>.

Radius dimension
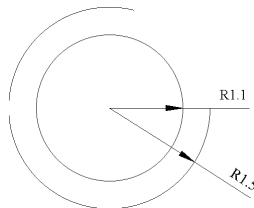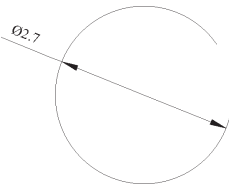
Diameter Dimension

Menu: Insert|Dimension|Diameter   **Toolbars**: Dimension, Insert

Use the Diameter Dimension tool to dimension the Diameter of an arc or circle.

1. Activate the Diameter Dimension tool.

2. Click on the arc or circle that you want to dimension. *KeyCAD* draws a dimension through the center of the circle or arc to the opposed points on its perimeter.

3. Define a second point to position the angle of the dimension and to specify the position of the dimension text. The text can be positioned either inside or outside the circle or arc.

Alternatively, you can click on the circle or arc that you want to dimension, specify the text, angle, and length of the dimension in the edit bar, then press <Enter>.

Diameter dimension

Leader Dimension

Menu: Insert|Dimension|Leader   **Toolbars**: Dimension, Insert

Use the Leader tool to create dimension text attached to a simple line leader pointing to a location in your drawing. The Leader tool is similar to the Multiline tools in that it can contain an arbitrary number of connected line segments, which can be oriented at any angle.

1. Define the starting point of the leader (the end that is pointing toward an object).

2. Define a second point to specify the second endpoint of the line segment. *KeyCAD* will draw a line between the two points with an arrowhead on the first point.

3. If necessary, define more points, adding new line segments to the leader.

4. Type the text of the note in the field on the Edit Bar.

5. Complete the leader by double-clicking, or choosing Finish from the local menu.

You can also use the Edit Bar to create a leader. First define the starting point of the leader, then enter the length and angle of the line segment in the Edit Bar. Press <Enter> to add a new line to the dimension without finishing the entity. Press <Ctrl>+<F> to complete the leader.

NOTE: *This tool serves a function similar to the Arrow tool in KeyCAD 2.0.*

use this line
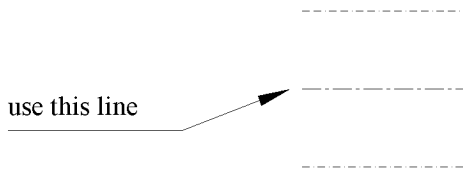
Leader dimension

Insert File

Menu: Insert|File   **Toolbar**: Insert

Use this to insert a *KeyCAD* or other CAD file into an open drawing:

1. Choose Insert|File to display the Insert File dialog.

2. Choose the type of CAD file you want to import from the List Files of the Type dropdown list.

3. If necessary, navigate to the directory where the graphic file you need is located.

4. Choose the file that you want to open from the File Name list box, then click the OK button.

The complete contents of the file will be inserted into your drawing, alongside any existing objects. The file types that can be inserted are the same as those that can be opened in the Open dialog.

TIP: *To see the entire file after it has been inserted, choose View|Zoom|Extents.*

SEE ALSO: *Supported File Types*

Insert Picture

Menu: Insert|Picture   **Toolbar**: Insert

Use this command to insert a picture into your drawing. The picture file can be of type BMP (bitmap). Once the image is inserted, you can move it, copy it, or scale it. You cannot, however, rotate pictures or edit their contents.

To insert a bitmap picture:

1. Choose Insert|Picture to display the Open dialog.

2. Choose the type of graphic file you want to import from the List Files of Type dropdown list.

3. If necessary, navigate to the directory where the graphic file you need is located.

4. Choose the file that you want to open from the File Name list box and click the OK button.

Tip: *The picture you insert is actually an OLE object, so you can double-click the picture in KeyCAD and then edit it using Paint (or whatever bitmap editor is registered in your Windows system).*

SEE ALSO: *Insert Object*

Insert Object

Menu: Insert|Object   **Toolbar**: Insert

The Insert Object command lets you insert OLE (Object Linking and Embedding) objects from other Windows applications into your *KeyCAD* drawing. You can edit the object from within *KeyCAD* using the features and commands of the Windows application in which the object was created (the *source* application). For example, you can insert a

Microsoft Word application into your drawing and edit the text or graphics using Microsoft Word tools.

To insert an OLE object into your drawing, first choose Insert|Object to display the Insert Object dialog. You can then Choose one of two basic options by clicking the Create New or the Create From File option button (Create New is the default). The following sections describe how to insert objects using these two options.

Adobe Acrobat file in KeyCAD

SEE ALSO: *Edit Links, Edit Object, Paste Special*

Create New

The Create New option lets you embed a new OLE object from another application into your drawing. You can edit the object using the tools of the source application.
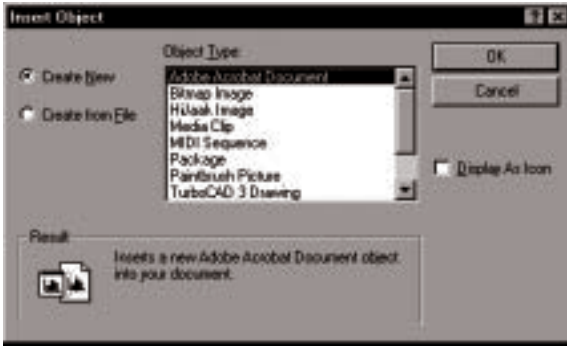
Insert Object dialog

1. Choose Insert|Object to display the Insert Object dialog.

2. If you want an icon representing the OLE object to be placed in your drawing, click the Icon check box.

3. Choose an OLE application from the dialog's scrolling list: Either double-click the application's name, or select it and press <Enter>. The object will appear in your *KeyCAD* drawing.

Create From File

The Create From File option lets you either embed or link an OLE object into your drawing based on a file previously created in the source application. See the following section on Embedded and Linked OLE Objects for an explanation of the difference between embedding and linking.
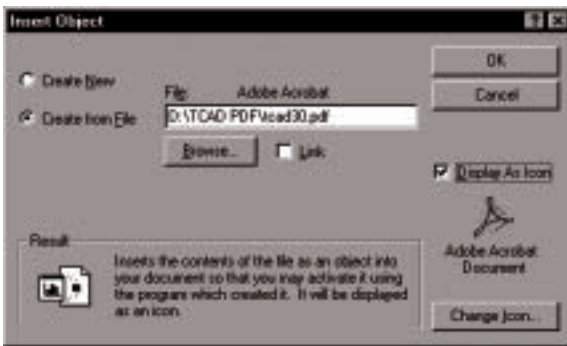
Insert Object dialog

1. Choose Insert|Object to display the Insert Object dialog.

2. Click the Create From File radio button. The appearance of the dialog will change.

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3. If you want the OLE object to be displayed as an icon in your drawing, click the Icon check box.

4. Check the Link check box if you want the OLE object to be linked; otherwise it will be embedded.

5. Type the path and name of the file that you want to use in the text box, or click the Browse button and locate the file using the Open dialog.

An OLE object based on the file will be inserted into your *KeyCAD* drawing. To edit the object in *KeyCAD* using the application that created the file, double-click the object, or select the object and choose Edit|Object. To update or edit links to a linked object, use the Edit|Links dialog.

SEE ALSO: *Edit Object, Edit Links, and Paste Special*

Embedded and Linked OLE Objects

OLE objects that you insert can be either *linked* or *embedded*. Note these important differences between embedded and linked OLE objects:

• Embedded objects become an integral part of your *KeyCAD* file, and take up more space in the file than a linked object. Embedded objects will move with the file if you transfer it to another computer system. If you make subsequent changes to the file on which the embedded OLE object was based, these changes will *not* affect the embedded OLE object.

• Linked objects are references to the file on which they were based, and they retain their connection to that file. If you make changes to the file on which a linked OLE object is based, the changes will also affect the linked OLE object.

SEE ALSO: *Edit Links*

WARNING: *If you move a KeyCAD file containing a linked object to another system, you will need to move any linked files along with the KeyCAD file. Linked OLE objects are referenced to their files by a specification of their drive and path. If you move the linked file to another drive or directory, Windows will attempt to find the linked file, but if it cannot locate the file, you will need to change the drive path and directory using the Edit|Links dialog.*

7
Formatting Objects

Format Menu and Tools

The Format menu contains commands and tools for setting the appearance and format of drawing entities and other objects. These features enable you to:

- Format objects by setting their properties.
- Combine objects into groups.
- Explode groups and objects into their constituent parts.
- Apply hatch patterns to objects.
- Send objects to the back and bring them to the front.
- Align multiple selected objects.

The first part of this chapter discusses some general concepts and techniques that will help you perform these tasks and describes how to format using the Properties and Text toolbars. The remainder of the chapter provides a complete reference to the commands and tools available from the Format Menu.

Setting Properties

Every object in *KeyCAD* has a set of *properties*. The exact set of properties depends on the type of object. All drawing entities share the following common sets of properties, organized by property sheet in the Properties dialog:

General. The layer on which the object resides, and a text attribute (generally its name) associated with it.

Pen. The color, width, and style of lines.

Brush. Determine the characteristics of fills and hatches.

There are also many other properties associated with specific classes of entities. Dimensions, for example, have a large number of properties that apply specifically to them, located on the Format, Advanced Format, and Advanced Text property sheets. Double lines, text, and curves also have properties and corresponding property sheets specific to them.

When you set properties, you can either set the default properties for a drawing tool, or set the individual properties of selected objects. These options are covered in the following two sections.

NOTE: *The Properties dialog is context-sensitive, meaning that it will display only the property sheets appropriate to the currently selected tool (or the currently selected objects). For example, if you select a line segment and a double line segment, then bring up the Properties dialog, you will see the General, Pen, Brush, and Double-Line property sheets. You won't see the Curve property sheet, for example, because the objects you have selected don't include any curves.*

SEE ALSO: *Properties Dialog*

Setting the Default Properties of a Tool

When you set the default properties of a tool, all subsequent entities drawn with that tool will have the settings you specified, until you explicitly change them. To access the default properties, either right-click on the tool icon, or activate the tool and then choose Format|Properties.

Setting the Properties of Selected Objects

The Properties dialog is context-sensitive, meaning that when you open it, it will display the set of property sheets that are appropriate to the objects selected. If you select a double line, or a set of entities that include a double line entity, then bring up the Properties dialog, the properties dialog will contain the Double Line property sheet.

You can use this dialog to set the default properties for a drawing tool; or to set the individual properties of a particular object or set of objects.

If you have property settings that you want to reuse, you can save them as a style, then reapply the style later to other entities.

SEE ALSO: *Layers Property Sheet, Properties Dialog, Property Toolbar*

Property Toolbar

The Property toolbar gives you a quick and convenient way of setting the properties of any selected object or set of objects, or for setting the default properties of a drawing tool. If the Property toolbar is not visible on the screen, you can display it using the View Toolbars dialog.
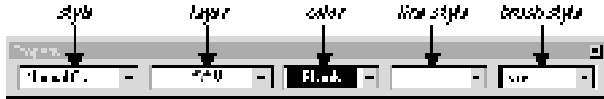
Property toolbar

If no objects are currently selected, the settings on the Property toolbar apply to the active drawing tool. For example, if you activate any single line tool and then change the settings on the Property toolbar, the settings will apply to all objects drawn with any single line tool.

If there are objects selected, settings in the Property toolbar apply to the selected objects.

NOTE: *In KeyCAD 3.0 you cannot apply a style to a selection. The Style pulldown box is accessible only when you are setting the default properties of a drawing tool.*

The Property toolbar consists of five pulldown list boxes. From left to right they are:

Style. Use this list box to apply predefined property settings to a drawing tool. You can save styles in the Properties dialog. This control can be used only to set the properties of a drawing tool; it is unavailable if you are setting the properties of selected objects.

Layer. Use this list box to set the layer on which objects are drawn.

Color. Use this list box to set the pen color.

Line Style. Use this list box to set the style of the line. You can use a continuous line or choose from several types of dotted or dashed lines.

Brush Style. Use this list box to choose a fill pattern for closed entities.

SEE ALSO: *Layers Dialog, Properties Dialog*

Formatting Text

You can format text in *KeyCAD* using either the Text Format toolbar or the Text property sheet. The Text property sheet, which offers more options than the Text Format toolbar, will be covered in the Command Reference section of this chapter.

You can display and hide the Text Format toolbar via the Desktop property sheet, which you can access by choosing either View|Toolbars or Tools|Desktop. You can use Text Format toolbar either to set default properties for the Text tool (which will be saved with your drawing) or you can use it to format selected text. To change default settings, activate the Text tool on the Insert toolbar, and then choose settings on the Text Property toolbar. These setting will stay in effect until you explicitly reset them. To format selected text, first select the text using the Select tool, then choose settings in the Text Format toolbar.

Text Format toolbar

The Text Format toolbar has the following controls:

Font. Pull down this list box to set the font. You can use any TrueType font that is installed on your system.

Size. Pull down this list box to set the text size. (The size can be in either World or Paper units, depending on the current mode. Check the icon in the upper-left corner of the ruler.)

Attributes. Click the Bold, Italic, and Underline buttons to turn these text attributes on and off.

Justification. Click these buttons to choose whether the text is left, center, or right-justified in its frame. Left-justified text will be typed in to the right of the point where you placed the text; right-justified text extends to the left of this point; center-justified text will be centered on the point.

SEE ALSO: *Manipulating Text, Text, Text Property Sheet*

Groups

KeyCAD drawing objects can be combined into groups. The drawing objects can be entities, such as lines circles and arcs, or they can be other groups. You can include groups in groups. *KeyCAD* treats a group as a single object for purposes of selecting and editing.

To break up a group into its constituent parts, select it, then use the Format|Explode command.

The names of groups are held in the Attribute field of the properties dialog. To view or edit this field, activate the Select tool, and then double-click on the group.

Because groups can themselves contain groups, you can think of them as complex *hierarchical* structures. This type of structure can be represented as a tree, as shown in the following diagram:

Structure of group represented as a tree

TIP: *You can view the structure of a group by selecting it and exploring the Selection Info palette (View/Selection Info).*

SEE ALSO: *Create Group, Explode*

Command Reference

The following table summarizes the contents of the Format menu:

Command	Function
Properties	Edit the properties of objects.
Create Hatch	Draw a hatch pattern over selected objects.
Create Group	Define all objects in the selection as a group.
Explode	Break up an object into its constituent parts.
Send To Back	Send selected objects to the back of the object stack on their layer.
Bring To Front	Bring selected objects to the front of the object stack on their layer.
Align	Line up all selected objects on the left, right, top, bottom, vertical center, horizontal center, or along a line that you specify.

Properties Dialog

Menu: Format|Properties   **Toolbar:** Modify

The Properties command displays the Properties dialog. Use this dialog to set the properties of objects, or the default properties of drawing tools.

The Properties dialog contains a set of property sheets. The General, Pen, and Brush property sheets are available whenever you display the dialog. Other property sheets (Double Line, Curve, Point, Text, and the three dimension property sheets) are displayed only when the tool to which they apply is activated, or if an entity to which they apply is selected.

SEE ALSO: *Select Commands, Property Toolbar*

General Property Sheet of Properties Dialog

The General property sheet of the Properties dialog lets you:

- Select the layer of the drawing on which objects are drawn.
- View or edit an object's *text attribute*.
- Select a style (a set of predefined property settings).
- Save a set of property settings as a style, or edit the property settings of an existing style.
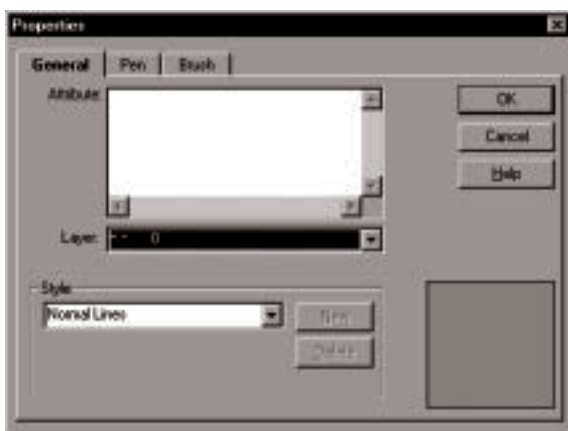
General property sheet of Properties dialog

The property sheet contains the following controls:

Attribute. To give selected objects a text attribute, click in the Attribute text box and type in whatever information you want to record. You might, for example, want to record the part number or cost of a physical component represented in the drawing. The Attribute text field is used to hold the names of groups.

Layer. To set the layer property, which determines the layer on which entities are drawn, click on the Layer drop-down list box, then click on the desired layer. You can create new layers using the Layers dialog (Tools|Layers).

Style. A style is a set of saved property settings. You can select and edit existing styles, or create new styles, as described in the following section.

Selecting, Editing, and Creating New Styles

NOTE: *Styles are applicable only if you are setting the default properties of a drawing tool. You cannot apply a style to selected objects, nor can you define a style based on the properties of selected objects.*

To select a style that has already been defined, click the down arrow on the right side of the Style text box and then click to select the style from the drop-down list that appears. The property settings that define that style will now become the default property settings of the tool, the properties of which you are editing.

To edit an existing style:

1. Select the style from the drop-down list.
2. Make changes to settings in the Properties dialog, in any of the property sheets that are available in the dialog.
3. Click the OK button to close the dialog.

To create a new style:

1. If you like, select a style that you want to use as a basis for a new style. (If you skip this step, then you will use whatever style is currently selected.)
2. Select a style name in the text box and type a name for the new style.
3. Click the New button. This creates a new style that has all the property settings of the style you selected in step 1.
4. Make any changes you like to settings in the property sheets, then click the OK button. The property sheet settings will be saved in the new style, which will be available the next time you open the Properties dialog for the tool for which you are setting properties.

Pen Property Sheet

The Pen property sheet lets you set properties of the drawing pen, such as its pattern, color, and thickness.
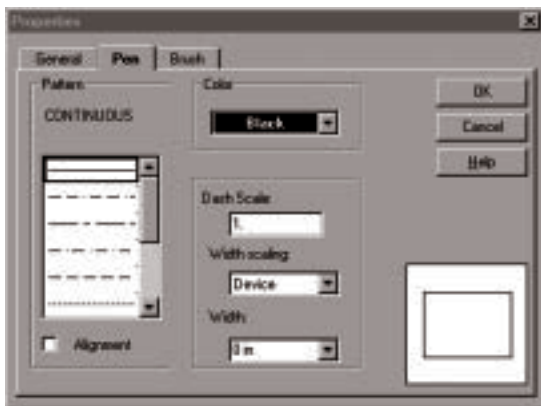
Pen property sheet

The Pen property sheet contains the following controls:

Pattern. Scroll through this list to choose a pattern for the line. You can choose either a continuous line or one of many dot-and-dash patterns.

Alignment. Click this box if you want the pattern to align itself so that the corners of rectangles and polygons are always solid.

Color. Click in this box to choose a color for the drawing pen. If you select By Layer, the pen color will be set to the default color of the object's layer. This color is set in the Layers property sheet.

NOTE: *You can create new colors using the Color Palette property sheet (Tools/Color Palette).*

Dash Scale. You can modify the factor in this box to set the scale of the pen's dot-and-dash pattern. For example, if you set the scale to .5 (instead of 1, the default), the dots and dashes will be half the default scale, and the pattern will therefore be repeated twice as often.

Width Scaling. This setting determines whether the width of the line will

be scaled, or remain the same size, when you zoom in and out of the drawing. If you select Device, the width will be scaled relative to your computer screen, and will remain the same size at any magnification. If you choose World, then the width will be scaled with your drawing and will change size as you zoom in and out. This setting also applies to the size of the elements in the pen's dot-and-dash pattern.

Width. Click in this box to choose a width for the pen, or type a custom width directly into the box. (To set the unit of measure for the Pen Width, use the Advanced Units property sheet.)

SEE ALSO: *Advanced Units Property Sheet, Layers Property Sheet*

Brush Property Sheet

The Brush property sheet lets you set fill properties of objects, as well as the properties of hatches. Fills appear only on closed entities, such as circles and polygons.
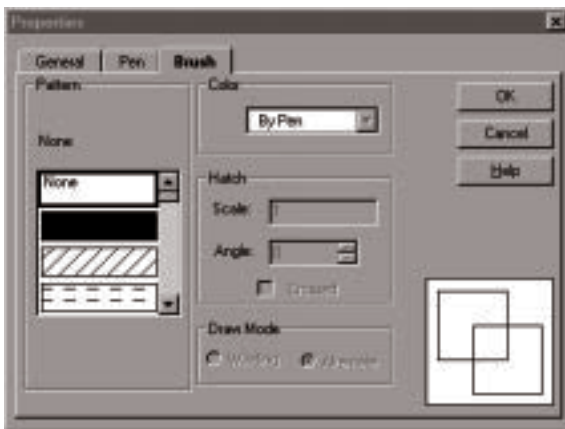
Brush property sheet

The Brush property sheet contains the following controls:

Pattern. To change the fill pattern, scroll through the Brush Style list box and click to select the pattern that you want. The name of the pattern will be displayed just above the Brush Style list box.

Color. To change the color of the fill, click on the Color drop-down list box, then scroll through the list and click to select the color that you want. Choose By Pen if you want the Brush color to match the Pen color. Choose By Layer if you want the color to match the default color of the object's layer (set in the Layers property sheet).

NOTE: *You can create new colors using the Color Palette property sheet (Tools/Color Palette).*

Hatch. Use the Hatch controls to control the scale and angle of the fill pattern. The default scale of the pattern is 1, so increasing it to 2 doubles the size of the pattern elements, and decreasing the scale to .5 halves the size of the components. Use the Angle spin button to change the angle of the hatch pattern, rotating it inside the filled entity (or type a custom value directly into the text box). If you check the Cross Hatch check box, the pattern will be drawn across the entity a second time, in a direction perpendicular to the first pattern. This gives you many more choices of hatch patterns.

Draw Mode. Choose an option button in this group to specify how hatch patterns are drawn over overlapping objects. Choose the Winding option if you want the hatch pattern to cover areas where filled entities overlap. Choose Alternate if you want the hatch pattern to be drawn on every other overlapping layer, creating an alternating pattern.

SEE ALSO: *Color Palette Property Sheet, Property Toolbar*

Curve Property Sheet

The Curve property sheet is visible only when a Bezier or spline curve entity is selected, or when the Bezier or spline curve tool is active and no entities are selected. This property sheet does not apply to sketch curves.
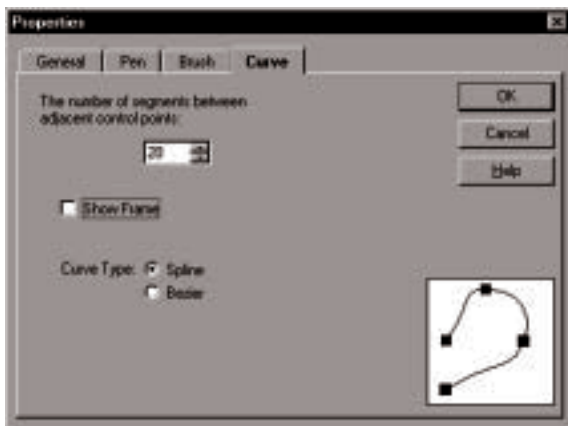
Curve property sheet

The Curve property sheet contains the following controls:

The number of segments between adjacent control points. Curves are composed of a number of minuscule line segments drawn between control points. Spline curves intersect the control points; Bezier curves gravitate toward the control points but do not intersect them. Use this spin control to set the number of line segments drawn between each control points. Large numbers will round the contours of the curve; small numbers will flatten out the contours.

Show Frame. Check the Show Frame check box to display a multiline frame connecting the curve's control points. This provides visual feedback about how the curve is drawn.
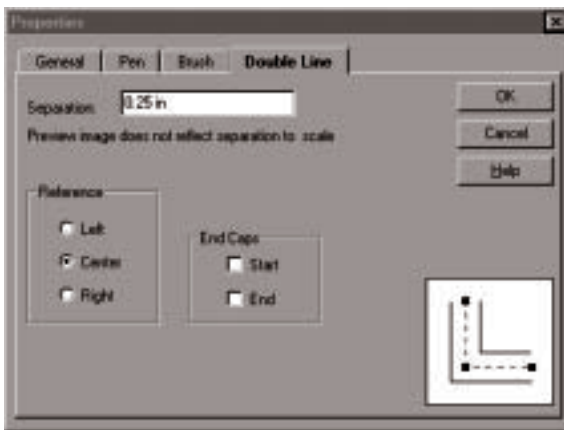
Curve Type. Use the Curve type option buttons to set whether the curve is drawn as a Bezier or spline curve.

SEE ALSO: *Curve Tools*

Double Line Property Sheet

The Double Line property sheet is visible only when a double line entity is selected, or when a double line tool is active and no entities are selected.

Double Line property sheet

The Double Line property sheet contains the following controls:

Separation. To set the distance between the two lines of the double line, specify the distance in the Separation text box.

Reference. Use the Reference option buttons to determine the alignment of the double line nodes. These points determine how the double line will align itself to coordinate positions when you draw or position the line. You can set the double line nodes to align to its left line, its right line, or to a centerpoint exactly between the two lines. (You can edit these nodes directly in Node Edit mode).

End Caps. Click in the Start and End check boxes to determine whether the startpoint and endpoint of double lines are open or closed.

SEE ALSO: *Double Line Tools, Node Edit of Double Lines*

Dimension Property Sheets

Before discussing how to format dimensions using the dimension property sheets, this section will describe the component parts of dimensions.

Dimensions consist of three basic components:

- The dimension lines, with arrows pointing to either end of the dimension.

- Extension lines, which connect the dimension line to the object being dimensioned.

- Dimension text, normally displaying the distance being dimensioned in World units.
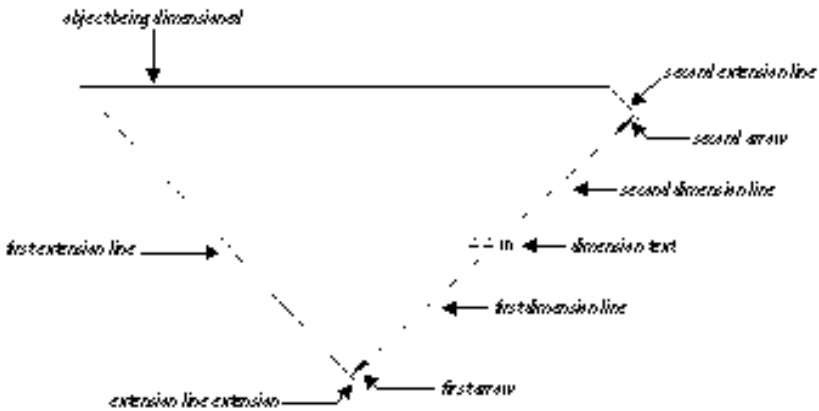
Parts of a Dimension

Extension lines have optional line segments that continue the extension outward beyond the dimension line (extensions to the extension).

A dimension can have interior dimension lines, as shown in the preceding figure, or two exterior dimension lines. Exterior dimension lines can be supplemented with an optional interior line.
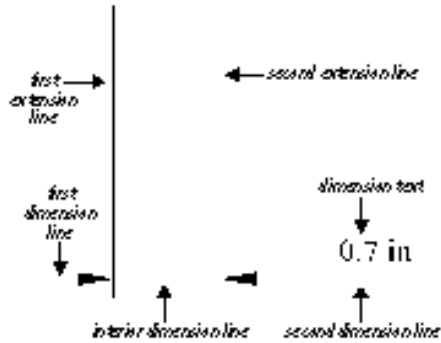
Dimension with Exterior Dimension Lines

The three dimension property sheets contain options that let you precisely format dimension lines, extension lines, and text:

Format. Lets you format the shape of dimension arrows and set the position of dimension text relative to the dimension line and extension lines.

Advanced Format. Gives you precise control over the drawing of dimension lines and extension lines.

Advanced Text. Gives you precise control over the appearance of dimension text.

The following sections describe these property sheets.

SEE ALSO: *Dimension Tools*

Format Property Sheet

The Format property sheet gives you control over the shape of dimension arrows and the position of text relative to the dimension line.

Format Property Sheet

The Arrowhead group contains the following controls:

1st and 2nd. Click in these drop-down boxes to choose dimension arrow shapes. The arrowheads can have angles of either 30 or 45 degrees; they can be filled, open, or closed. If you like, you can choose None (no arrow at all), or Tick (a simple angled line).

Size. Click in this control to choose a size for the length of the arrowhead, or type in a custom size.

Tick Extension. If you set the arrowhead to Tick in the 1st and 2nd boxes, you can specify the length of the tick's extension line in the Tick Extension control.
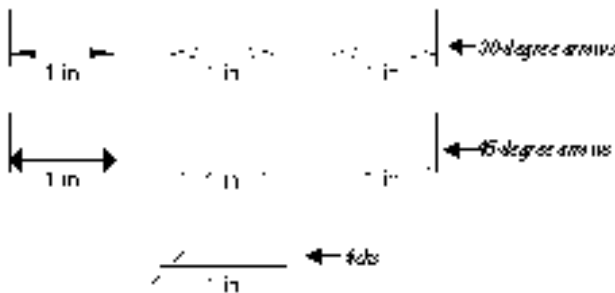
Dimension arrow shapes

The Text Position group contains the following controls:

Font. Click in this box to choose a TrueType font for the dimension text. All TrueType fonts installed on your system will be available.

Height. Click in this text box to choose a text height, or type in a custom height. You can set the height in either World or Paper units, depending on the current mode.

Position. Choose Outside if you want the text placed outside of the dimension line, Above if you want the text "above" the dimension line ("above" is relative to the orientation of the text, not relative to your screen), or On Line if you want to place the text directly on the dimension line. This position can be manually adjusted (see the next control).

On Line Adjustment. Type a number into this text box to specify the distance of the text above or below the dimension line. A value of zero (the default value) will place the text on the same level as the dimension line. Higher values move the text above the line ("above" is relative to the orientation of the text); lower (negative) values move the text below the line.

Force Text Horizontal. Check this box if you want the dimension text to be horizontal.

Force Dimension Text Inside Extension Lines. Click this box if you want to keep dimension text between the extension lines, regardless of the distance between the extension lines. If you leave this box unchecked, *KeyCAD* will decide whether to place dimension text inside or outside the extension lines.
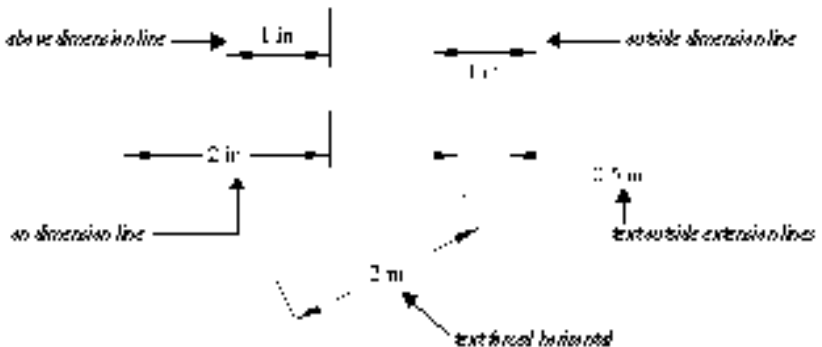
Various positions for dimension text

Advanced Format Property Sheet

The Advanced Format property includes options for drawing the dimension and extension lines, as well as an option for scaling the size of dimensions relative to your drawing. If you're not sure what any of the terms mean in this section, read the section "Parts of a Dimension" earlier in this chapter.
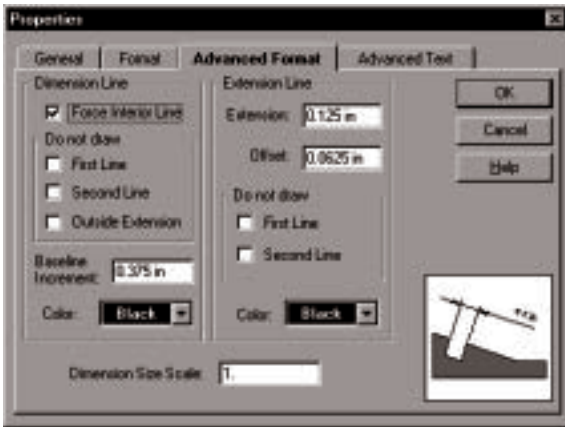

Advanced Format property sheet

The Dimension Line group includes the following controls:

Force Interior Line. Check this box if you want the dimension to have an interior dimension line even if the text is outside the extension lines. If you leave this box unchecked, *KeyCAD* will draw the interior dimension line only if the dimension text is inside the extension lines.

Do Not Draw. Click First Line if you don't want *KeyCAD* to draw the first dimension line; click Second Line if you don't want *KeyCAD* to draw the second dimension line.

Color. Click in this box and choose a color if you want the dimension lines to have a different color to distinguish them from the other parts of the dimension.

The Extension Line group includes the following controls:

Extension. The number in this box determines the length of the extension line segments that extend outward beyond the dimension lines. You can enter a value in the box to specify the length of these segments.

Offset. Type a value in this box if you want to specify the distance between the extension lines and the object that is being dimensioned.

Do Not Draw. Click First Line if you don't want *KeyCAD* to draw the first extension line; click Second Line if you don't want *KeyCAD* to draw the second extension line.

Color. Click in this box and choose a color if you want the extension lines to have a different color to distinguish them from the other parts of the dimension.

At the bottom of the property sheet you will find a *Dimension Size Scale* text box. This box controls the size of the dimension text and arrows relative to the rest of your drawing. If you use the default setting of 1, the dimension text will be printed at the size specified in the Height box on the Format property sheet.

TIP: *Choose a smaller scale if you want to dimension details in the interior of your drawing.*

SEE ALSO: *Parts of a Dimension*

Advanced Text Property Sheet

The Advanced Text property sheet contains controls for precisely formatting the appearance of dimension text, and for controlling the display of tolerances. (Tolerances are a number indicating the amount of allowable deviation from the distance shown in the dimension.)
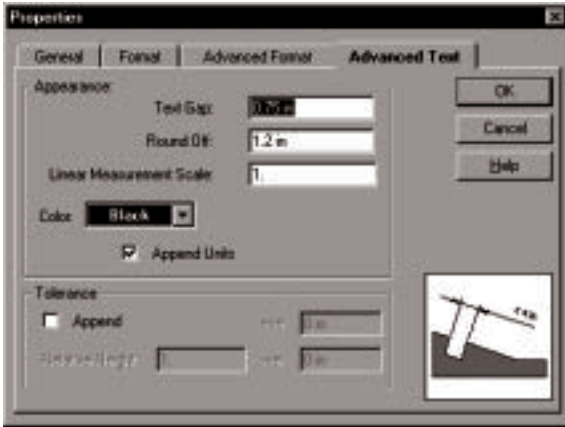
Advanced Text property sheet

The Appearance group contains the following controls:

Text Gap. Type a number in this box if you want to specify the distance between the dimension text and the dimension line.

Round Off. Type a number in this box to indicate the decimal place to which the dimension text will round off. If you type a value of 0.1, for example, the text will be rounded off to tenths of a unit.

Linear Measurement Scale. The default value for linear measurement scale is 1, which means, for example, that a dimension will display 1 inch if it is dimensioning a distance of 1 inch in World units. The Linear Measurement Scale control allows you to change the scale of the value displayed by the dimension text relative to World units: If you enter a value of .1, for example, the dimension will display a value of .1 inch when dimensioning a distance of 1 inch in World units.

TIP: *You will probably want to leave Linear Measurement Scale unchanged, unless you have an inset in your drawing that uses a different scale than the rest of your drawing.*

Color. Click in this box to choose a color for the dimension text.

Append Units. Check this box if you want the dimension text to display its unit of measure.

The Tolerance group contains the following controls:

Append. Check this box if you want the dimension text to include a tolerance. The other controls in the group are available only if this box is checked.

Relative Height. Type a value in this box to set the height of the tolerance relative to the rest of the dimension text. If you type a value of .5, for example, the tolerance will be half the height of the rest of the dimension text.

+ve. Type a value in this box to set the amount of positive tolerance.

-ve. Type a value in this box to set the amount of negative tolerance.

Point Property Sheet

The Point property is included in the properties dialog only when a point entity is selected, or when a point tool is active. It includes settings for changing the size of a point, its shape, and its color.
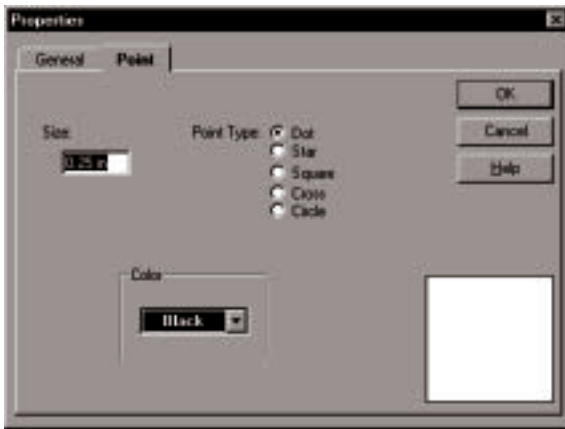
Point property sheet

The Point property sheet contains the following controls:

Size. Type a value in this box to set the width and height of the point. This value affects all types of points except the dot, which is always displayed at a minimal size. You can set the size in either World or Paper units, depending on the current mode. (Check the icon at the upper-left corner of the ruler to see whether *KeyCAD* is showing World or Paper units.)

Point Type. Click in an option button to select the type of point (dot, star, square, cross, or circle).

Color. Click in this box and scroll to choose a color for the point from the color palette.

Text Property Sheet

The Text property sheet is included in the Properties dialog only when a text entity is selected, or when the text tool is active. It includes options for setting font, style, height, color, text effects, and text justification.

Text property sheet

The Text property sheet contains the following controls:

Font. Click on this box and scroll to select a TrueType font for the text. All the TrueType fonts that are installed in your system will be available.

Height. Click in this box and scroll to choose a text height, or type a custom text height directly into the box. You can set the height in either World or Paper units, depending on the current mode. (Note that text can be measured in points only if you are using Paper units.)

Style. Click in this box to choose a text style (Normal, Italic, Bold, or Bold Italic).

Color. Click in this box and scroll to select a color for the text from the color palette.

Effects. Check the options in this box to turn on any of these text effects: Underline, Strikethrough, Text Box (a simple border around the text), Hidden (used for notes that you don't wish to display), and All Caps.

Justification. Click an option button to choose whether the text is left-, center-, or right-justified. When you are creating text, left-justified text is typed to the right of the insertion point; right-justified text is typed to the left of the insertion point, and center-justified text is centered on the insertion point.

NOTE: *If you change the justification of existing text, the change will have an effect only if the existing text has multiple lines.*

TIP: *You can get more precise control of the placement of text relative to the insertion point by using the Align feature. You can access this feature by right-clicking in the drawing area while the Text tool is active, and then choosing the Align... command from the local menu. The Align feature can only be applied to text as it is being drawn; you can't use it to format existing text.*

Create Hatch

Menu: Format|Create Hatch **Toolbar:** Entity Format

The Create Hatch command applies a hatch pattern to selected entities.

NOTE: *For you to choose a hatch pattern, the Entity Format toolbar must be displayed on screen. To display this toolbar, choose View|Toolbars, check Entity Format in the Toolbars scrolling list, then press <Enter> to close the dialog.*

1. To set the hatch pattern you want to use, right-click on the Hatch tool to bring up the Properties dialog, then click the Brush tab to bring the Brush property sheet to the front.
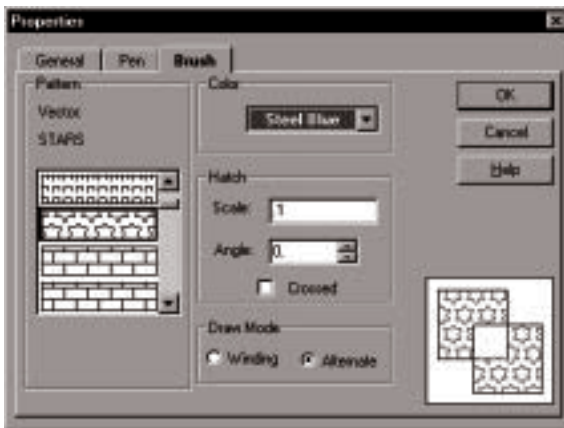
Brush property sheet

2.   Use the Brush property sheet to format the hatch pattern. Formatting a hatch pattern is essentially the same as formatting a fill pattern, so you can refer to the Brush Property sheet section earlier in this chapter for a description of the property sheet options.

3.   Use the Select tools to select the object or objects that you want to hatch.

4.   Click the Hatch tool (or choose Format|Hatch) to apply the hatch pattern to the selection.

NOTE: *A hatch pattern, unlike a fill pattern, is an object independent of the objects to which it has been applied. You can select the hatch and manipulate it independently.*

SEE ALSO: *Brush Property Sheet*

Create Group

Menu: Format|Create Group

Use the Create Group command to combine any currently selected drawing objects into a group.

1.   Use any selection method to select a set of drawing objects. You can select drawing entities or groups.

2.   Click the Create Group tool or choose Format|Create Group from the menu.

SEE ALSO: *Groups, Explode*

Explode

Menu: Format|Explode

Use the Explode command to break up any object into its constituent parts. You can explode an object or group of objects by selecting them, then choosing Format|Explode.

When you explode an object, you break it down to the next level of constituents. For example, you might explode a group that contains a group, a line segment, and some text. After exploding the group, you could

then explode its constituents: the group into its parts and the text into polylines.

TIPS: *You can use Explode to break hatch patterns into constituent entities, and to break text into polylines.*

You can also explode double-line entities into groups of single-line entities. This is useful if you would like to scale the width of double lines along with an object that contains the double lines.

SEE ALSO: *Create Group, Groups*

Bring to Front and Send to Back

In a *KeyCAD* drawing, objects on each layer are stacked in the order that you create them, from back to front. The object at the front of the stack is visible on top of all other objects on its layer. The object furthest to the back of the stack is overlapped by all other objects on its layer.
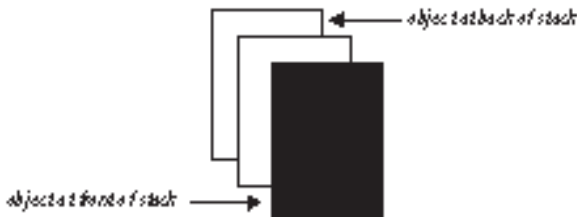
A stack of overlapping entities

KeyCAD provides two commands, Bring to Front and Send to Back, that will change the order of the entities in the stack, bringing the selection to the front, or sending it to the back.

WARNING: *The Bring to Front and Send to Back commands will not function as you expect if the objects are on different layers and the layers have different redraw order values. The redraw order value of each layer is set in the Layers property sheet. Layers with higher redraw order values will always be in front of layers with lower redraw order values. If the objects are on different layers and the layers have equal redraw order values, then Bring to Front and Send to Back will work across layers.*

TIP: *If you want to systematically place some categories of objects in front of others, use the redraw order property of layers.*

SEE ALSO: *Layers Property Sheet*

Bring to Front

Menu: Format|Bring to Front

Use the Bring to Front command to bring any currently selected entities to the front of the entity stack. This command is especially useful when you are dealing with overlapping filled entities, or bitmaps.

1. Select the object or objects, using any selection method, that you want to bring to the front of the entity stack.

2. Choose the Bring to Front command.

Send to Back

Menu: Format|Send to Back

Use the Send to Back command to send any currently selected entities to the back of the entity stack. This command is especially useful when you are dealing with overlapping filled entities, or bitmaps.

1. Select the object or objects, using any selection method, that you want to send to the back of the entity stack.

2. Choose the Send to Back command.

NOTE: *Sending an object to back has no effect on its layer, and changing its layer has no effect on its position in the entity stack.*

Align Commands

Use the Align commands to line up any currently selected objects. With all the Align commands except Along Line, the objects are lined up relative to the bounding box of the selection. The *bounding box* is a virtual rectangle that completely contains all of the selected objects. (Because *KeyCAD* draws a frame around the objects you have selected, you can see the extents of the bounding box.)
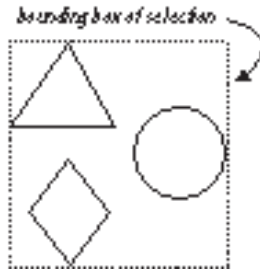
Bounding box of a selection

The following table summarizes the Align commands:

This Command	Aligns Objects
Align Bottom	Along the bottom of the bounding box.
Align Middle	Along a virtual horizontal line passing through the vertical center of the bounding box.
Align Top	Along the top of the bounding box.
Align Left	Along the left edge of the bounding box.
Align Center	Along a virtual vertical line passing through the horizontal center of the bounding box.
Align Right	Along the right edge of the bounding box.
Align Along Line	Along a construction line at a position and angle that you specify.

Align Bottom

Menu: Format|Align|Bottom   **Toolbar:** Entity Format

Use Align Bottom command to align the bottom edges of the currently selected objects along the bottom of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. *KeyCAD* will create a rectangular frame around the selection.

2. Choose the Align Bottom command from the menu or toolbar.

Align Middle

Menu: Format|Align|Middle   **Toolbar:** Entity Format

Use the Align Middle command to center the selected objects vertically in the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. *KeyCAD* will create a rectangular frame around the selection.

2. Choose the Align Middle command from the menu or toolbar.

middle of bounding box

Middle alignment

Align Top

Menu: Format|Align|Top   **Toolbar:** Entity Format

Use the Align Top command to align the top edges of the currently selected objects along the top of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. *KeyCAD* will create a rectangular frame around the selection.

2. Choose the Align Top command from the menu or toolbar.

Align Left

Menu: Format|Align|Left   **Toolbar:** Entity Format

Use the Align Left command to align the left edges of the currently selected objects along the left edge of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. *KeyCAD* will create a rectangular frame around the selection.

2. Choose the Align Left command from the menu or toolbar.

Align Center

Menu: Format|Align|Center   **Toolbar:** Entity Format

Use the Align Center command to center the currently selected objects at the horizontal center of the selection's bounding box.

1. Select all the objects that you want to align, using any selection method. *KeyCAD* will create a rectangular frame around the selection.

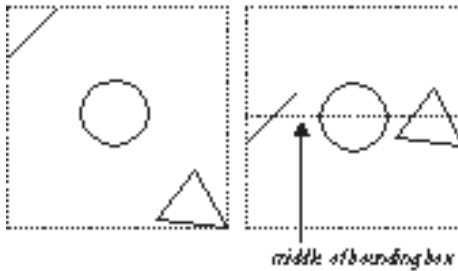2. Choose the Align Center command from the menu or toolbar.
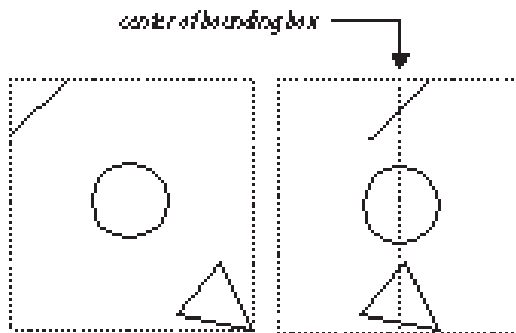
Center alignment

Align Right

Menu: Format|Align|Right   **Toolbar:** Entity Format

Use the Align Right command to align the right edges of the currently
selected objects along the right edge of the selection's bounding box.

1.   Select all the objects that you want to align, using any selection
     method. *KeyCAD* will create a rectangular frame around the
     selection.

2.   Choose the Align Right command from the menu or toolbar.

Align Along Line

Menu: Format|Align|Along Line   **Toolbar:** Entity Format

The Align Along Line tool differs from the other Align tools, in that it
aligns the selected objects relative to a line that you place at any position
an angle in the drawing space.

1.   Select all the objects that you want to align, using any selection
     method. *KeyCAD* will create a rectangular frame around the
     selection.

2.   Choose the Align Along Line tool from the menu or toolbar.

3.   Click the mouse at the point where you want to place the virtual
     line. *KeyCAD* will draw a line through the point of the mouse
     click.

4.   Define a second point for the line to pass through: The two
     defined points determine the angle of the line.

The virtual line disappears as the selected objects are aligned along it.
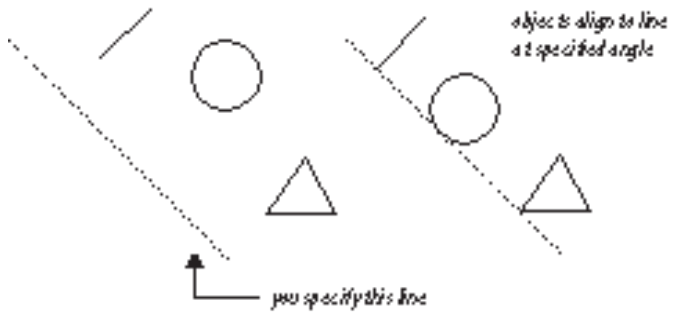
Aligning objects along a line

8
Geometric Editing

Modify Menu and Tools

The tools on the Modify menu are conventionally called *CAD editing* tools. These tools provide a variety of special editing functions for modifying the shape of entities. The following table summarizes the modify menu and its functions.

Command	Function
Split	Splits an entity into two entities.
Meet 2 Lines	Shrinks or extends two lines or double lines so that two of their endpoints form a clean intersection.
Chamfer	Joins two line segments or double line segments with a flattened corner.
Line Length	Changes the length of a line while retaining its connection with other lines in a polygon or multiline.
Shrink/Extend Line	Changes the length of a line so that it precisely meets another entity.
Fillet	Connects two lines, either single or double lines, with a smoothly fitted arc.
T-Meet 2 Double Lines	Forms a clean T intersection of two double lines.
Intersect 2 Double Lines	Forms a clean X intersection of two double lines.

NOTE: *There is practical distinction between the tools and the Modify menu and those on the Edit menu. When you use the Edit menu tools, you must first select the object or objects that you want to edit. The Modify menu tools are similar to the drawing tools, in that you first activate the tool, and then follow a series of steps to complete the edit. When you finish the edit, the tool remains active, so that you can either begin a second editing operation of the same type, or switch to another tool.*

Split

Menu: Modify|Split   **Toolbar:** Modify

Use the Split tool to divide an entity into two separate entities. This tool works on any single-line or double-line entity, as well as on circles, and arcs.

NOTE: *You can also use this tool to split curves and other complex entities, but you need to explode the entity first.*

SEE ALSO: *Explode*

Splitting Line and Double Line Entities

You can use the Split tool to split entities into two parts.

1. Activate the Split tool.
2. Click on the line or double-line that you want to split: Let's call this the *target*. *KeyCAD* draws a perpendicular line intersecting the target.
3. Move the mouse cursor until the perpendicular line is at the point where you want to split the target.
4. Click to split the target.

NOTE: *If the target is a closed entity (a rectangle or polygon), then you will need to split the entity in two places to divide it into two entities.*
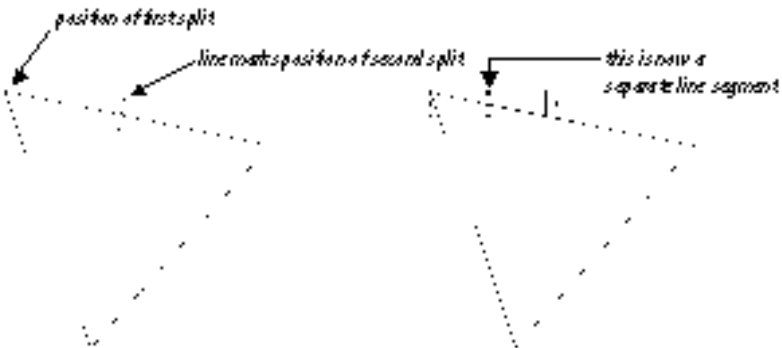
Splitting a polygon requires two splits

Splitting Arcs and Circles

To divide an arc into two arc segments:

1. Choose the Split tool.

2. Click on the arc that you want to split. A dotted line appears, connecting the cursor to the centerpoint of the arc.

3. Click to define the angle at which to split the arc.

Splitting a circle is the same as splitting an arc, except that at step 3 you need to define a starting and an ending angle (breaking the circle in two places).

Splitting an arc

Meet 2 Lines

Menu: Modify|Meet 2 Lines   **Toolbar:** Modify

Use the Meet 2 Lines function to shrink or extend two lines or double lines so that two of their endpoints meet. This function automatically cleans up the corner formed by two double lines.

1. Activate the Meet 2 Lines tool.

2. Click on the first, then the second line or double line.

When the two lines are crossed, the two lines will be shrunk in the direction of the mouse clicks. In other words, the two parts of the line that you click on will be preserved; the parts of the lines on the opposite side of the intersection will be trimmed.
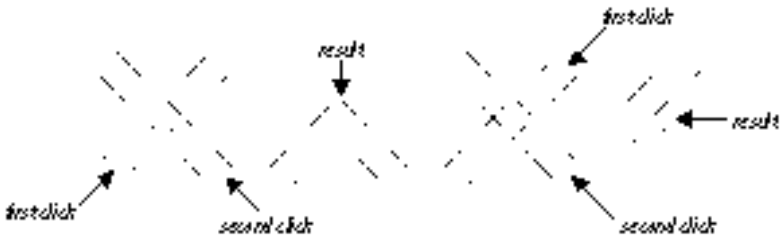
Using Meet 2 Lines

Chamfer Tools

Menu: Modify|Chamfer   **Toolbar:** Modify

Use the Chamfer tools to join two line segments or double line segments with a flattened corner.

Values for Calculating Chamfers

When you activate a chamfer tool, the values you can use to calculate the chamfer appear in the edit bar.

The following table describes these values and their meaning. Note that "point of intersection" means the point where the two lines intersect, or the point where they would intersect if they were extended.

Value	Meaning
Distance A	The distance from the point of intersection to the chamfer on the first line clicked.
Distance B	The distance from the point of intersection to the chamfer on the second line clicked.
Length	The length of the chamfer line.
Angle	The angle of the chamfer line from the first line clicked.

The following figure shows how these measurements on a chamfer:
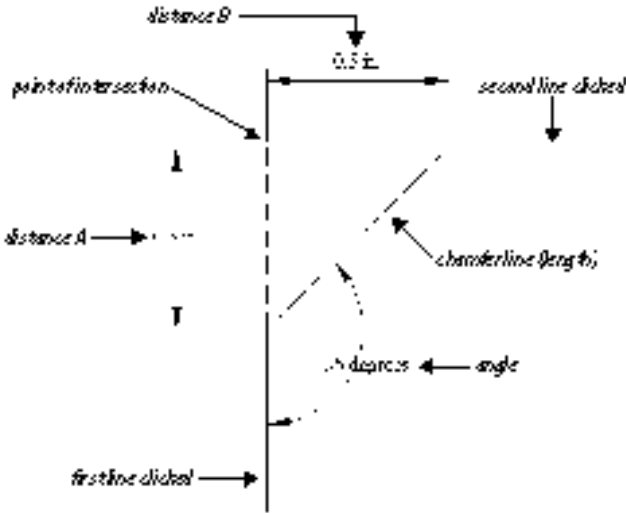
Values used in calculating chamfers

The *KeyCAD* chamfer tools use two combinations of these values:

- Distance and Distance
- Length and Angle

The following two sections discuss the specific tools for creating chamfers.

NOTE: *You can use the Multiline option to chamfer all the corners of a multiline or polygon simultaneously. The following sections provide instructions on how to do this.*

Distance and Distance Chamfer

Menu: Modify|Chamfer|Distance/Distance **Toolbar:** Modify

The Distance and Distance Chamfer tool draws a chamfer between points on each line calculated by their distance from the point of intersection on the first (Distance A) and second (Distance B) lines clicked:

1. Activate the Distance and Distance chamfer tool.
2. Enter values for distances in the Dist. A and Dist. B fields of the Edit Bar. Dist. A is the distance on the first line clicked. Dist. B

is the distance on the second line clicked. Return to the drawing area by clicking there or pressing <Ctrl>+<D>.

3.   Do one of the following:

•   Click on the first, then the second line.

•   Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will chamfer all the corners of the multiline or polygon that you selected.

Length and Angle Chamfer

Menu: Modify|Chamfer|Length/Angle   **Toolbar:** Modify

This method joins two lines with a chamfer of the specified length, at a specified angle from the first line clicked:

1.   Activate the Length and Angle Chamfer tool.

2.   Type values into the Angle and Length fields of the Edit Bar. Return to the drawing area by clicking there or pressing <Ctrl>+<D>.

3.   Do one of the following:

•   Click on the first, then the second line.

•   Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will chamfer all the corners of the multiline or polygon that you selected.

Line Length

Menu: Modify|Line Length   **Toolbar:** Modify

Use the Line Length tool to extend or shrink a line segment from either endpoint. If the line segment is part of a multiline or polygon, as you change the length of the line segment, it retains its relationship to other lines within a multiline or polygon. This means that lines connected to the line endpoint you are moving will also change angle and length so that they remain connected to the endpoint.

1.   Activate the Line Length tool.

2.   Click on a line segment at a position near the endpoint that you

want to modify. The segment now extends from the endpoint furthest from the mouse to the mouse cursor.

3. You can now set the line length in one of three ways:

- Enter an absolute length for the line in the Length field of the edit bar, then press <Enter>.

- Enter the amount that you want to lengthen the line in the Delta L field of the edit bar (enter a negative value to shorten the line), and then press <Enter>.

- Drag the mouse to lengthen or shorten the line, then click when it is the length you want.

Changing the length of a line segment in a polygon

Shrink/Extend Line

Menu: Modify|Shrink/Extend Line   **Toolbar:** Modify

Use the Shrink/Extend to Object editing tool to edit the length of a line so that it precisely meets another entity. You can either extend the line, increasing its length until it meets the other entity; or you can shrink the line, decreasing its length until it just meets the other entity. To use this tool:

1. Activate the Shrink/Extend to Object tool.
2. Click on the line or double line that you want to shrink or extend.
3. Click on the entity to which you want to shrink or extend the line. *KeyCAD* will adjust the length of the line or double line until it precisely meets the second entity.

extend this line to meet object

shrink this line to meet object

result of shrink or extend

Shrinking or extending a line to an object
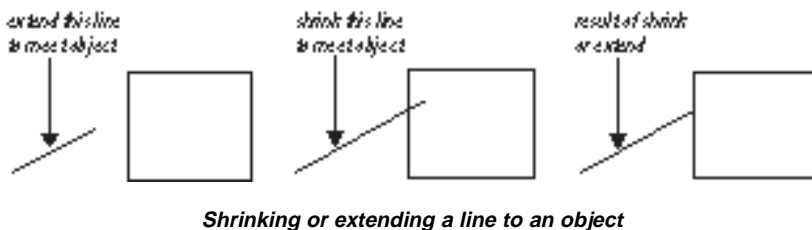
Fillet

Menu: Modify|Fillet   **Toolbar:** Modify

Use the Fillet tool to connect two lines, either single or double lines, with a smoothly fitted arc. The result is a rounded corner.

KeyCAD inserts an arc of a specified radius between two intersecting lines, making the arc tangent to both lines. The start and end angles for the arc are determined by the points at which the arc is tangent to the two lines. *KeyCAD* then trims or extends the lines to form the smooth corner.

NOTE: *You can use the Multiline option to fillet all the corners of a multiline or polygon simultaneously.*

1. Activate the Modify|Fillet tool.

2. Enter the radius for the filleted arc in the Radius text box on the Edit Bar and press <Enter>. (If the edit bar is not visible, display it using View|Toolbars.)

3. Do one of the following:

    •   Click on the first, then the second line.

    •   Click on a multiline or polygon, right-click to open the local menu, then choose Multiline. This will fillet all the corners of the multiline or polygon that you selected.
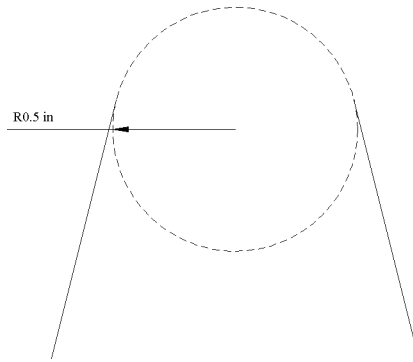
Radius of a fillet

NOTE: *If the radius specified is too large for the arc to fit between the two lines, or if you specify a radius of zero, KeyCAD will be unable to perform the fillet and will display an error message.*

T-Meet 2 Double Lines

Menu: Modify|Length   **Toolbar:** Modify

Use the T-Meet 2 Double Lines tool to form an T intersection of two double lines. The first line clicked will extend or shrink to form the stem of the T.

1. Activate the T-Meet 2 Double Lines tool.
2. Click on the line that will form the stem of the T.
3. Click on the line that will form the top of the T.

NOTE: *You can choose whether or not to make a clean intersection of the two double lines. To do this, activate the T-Meet 2 Double Lines tool, right-click in the drawing area, then check or uncheck the Cleanup option in the local menu.*

A clean T intersection of two double lines

Intersect 2 Double Lines

Menu: Modify|Length **Toolbar:** Modify

Use the Intersect 2 Double Lines tool to clean up the intersection of two double lines.

1. Activate the Intersect 2 Double Lines function.
2. Click on the first, then the second of two intersecting double lines.

NOTE: *You can choose whether or not to make a clean intersection of the two double lines. To do this, activate the Intersect 2 Double Lines tool, right-click in the drawing area, then check or uncheck the Cleanup option in the local menu. If you choose not to make a clean intersection, there will be a gap in the first line clicked, but not in the second line clicked.*
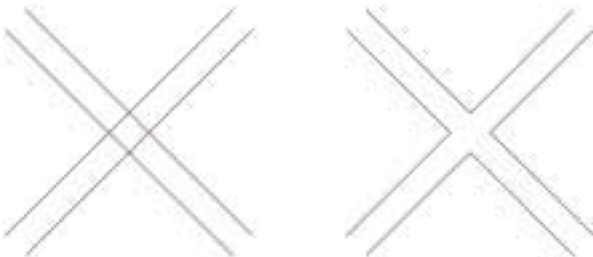
A clean X intersection of two double lines

9

Snaps and Coordinates

Modes Menu and Tools

Modes enable you to control how *KeyCAD* responds to your actions when you are drawing or editing. Some modes are mutually exclusive; others can be used in combination. This provides you with an enormous number of possible techniques, but the fundamental principles underlying all the modes are consistent and relatively simple. Once you understand these principles, you will be able to exploit them to find efficient and precise methods of drawing and editing.

There are two major categories of modes:

- *Snap modes*, which determine where exactly *KeyCAD* defines a point when you click the mouse in the drawing area.
- *Coordinate system modes*, which determine how you can use mathematical coordinates to specify the location of any point on the drawing.

In this chapter we'll first go over the general principles of using snap modes and coordinate system modes. The remainder of the chapter is a comprehensive reference to the modes and commands accessible from the Modes menu.

Snap Modes

Snap modes let you define points relative to objects in the drawing area when you are drawing or editing with the mouse. You can define points relative to drawing entities, construction entities, and the grid. All snap mode functions are similar: You first set the snap mode, then click on or near the object that you want to snap to. Rather than defining the point exactly where you clicked the mouse, *KeyCAD* will define the point precisely on the object. The point that *KeyCAD* chooses depends on the snap mode. You can, for example, snap to the end of a line, the midpoint of a line, the center of a circle, or a point on the grid.

Snap Aperture

The critical distance between the point of the mouse click and an object or grid point is the *snap aperture*. You can think of the snap aperture as a circular area centered on the point of the mouse click. If an object containing a qualifying snap point is located within the snap aperture at the time of the mouse click, your point will be defined at the nearest qualifying snap point. If more than one object is located within the snap aperture, your point will still be defined at the closest qualifying snap point.

Note that in most cases the snap point itself does not need to be located within the snap aperture. (Snap to Grid and Snap to Nearest on Graphic are obvious exceptions to this.)

TIPS: *To see the snap aperture while you are working, choose View|Aperture.*

To get additional visual feedback when using snap modes, turn on the Magnetic Point tool. When Magnetic Point is activated, the rubber band line that provides you feedback about the entity you are creating is attracted to the snap point. This lets you see exactly where the point will be defined after you click the mouse.
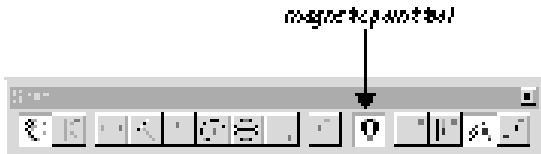
NOTES: *You can set the radius of the snap aperture in the General property sheet of the Tools|Program Setup dialog.*

Defining points with the status bar overrides the current snap mode and defines the point at precise coordinates that you specify.

SEE ALSO: *Program Setup Dialog*
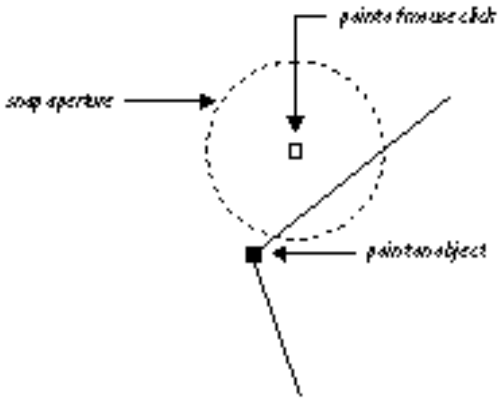
Use snap modes to define a point on an object near the point where you click the mouse

Local Snap

You can change snap modes in three different ways. You can use the menu, the Snap toolbar, or the local menu. When you choose a mode from the menu or the Snap toolbar, the effect is the same: The snap mode changes until you reset it. The local menu is different. Use it for "one shot" mode changes, or *local snap*. This means that when you change snap modes in the local menu, the mode remains in effect until you define the next point. *KeyCAD* then switches back to the last mode you set in the main menu or toolbar, or defaults to No Snap mode.

This is a useful feature, because it lets you set the snap mode you use most frequently in the main menu or toolbar, then switch modes "on the fly" in the local menu when you are working in your drawing. Many users, for example, like to work with Snap to Grid as their default snap mode, so they set the mode to Snap to Grid in the main menu. While working in a drawing they may need to snap, for example, to the corner of a polygon. To do this, they change the snap mode temporarily to Vertex in the local menu and define a point on the polygon corner. *KeyCAD* then automatically switches back to Snap to Grid, the mode they most commonly use.

Coordinate Systems

Coordinate systems provide different methods of specifying the location of any point in the drawing using coordinates. *KeyCAD* has two coordinate system modes: absolute and relative. Both the absolute and the relative coordinate modes are in the Cartesian coordinate system.

Cartesian Coordinates

A *Cartesian coordinate* specifies the distance of a point from the X or Y axis along a line that is parallel to the other axis. (When the drawing area is labeled with Cartesian coordinates it is a *Cartesian plane*.) You can specify the location of any point on the drawing using two coordinates, the first being the distance from the Y axis on a line parallel to the X axis, the second being the distance from X axis on a line parallel to the Y axis. The intersection of the X and Y axes is the *origin* of the coordinate system; its coordinates are (0,0).

For example, to specify the location of a point that is 2 units measured horizontally from the Y axis and 3 units measured vertically from the X axis, you would use the coordinates (2,3). For locations to the left of the Y axis or below the X axis you need to use negative values. To specify a location 2 units to the left of the Y axis and 2 units below the X axis, you would use the coordinates (-2,-2).
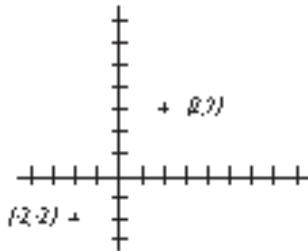
Points defined in Cartesian coordinates

Absolute Coordinates

When using absolute coordinates in *KeyCAD*, you specify the location of points relative to a location called the Absolute Origin, point (0,0) in the drawing plane. The Absolute Origin never changes location. By default, this point is in the lower-left corner of the page (although you can change the position of the page using the Page Setup dialog).

SEE ALSO: *Page Setup Dialog*

KeyCAD for Windows

Relative Coordinates

When you are working in relative coordinates in *KeyCAD*, you specify points on the drawing relative to a point that we'll call the Relative Origin. This point moves in the drawing space as you work. Suppose, for example, that you are drawing a multiline. As you define each point of the multiline, the point you just defined becomes (0,0). This enables you to draw each segment of the multiline relative to the last segment.

KeyCAD also has a feature that lets you fix the Relative Origin in one place, then move it manually. If you check the Fixed Relative Origin box in the Program Setup dialog, the Relative Origin will stay in one place until you explicitly move it, using the Relocate Origin command.

SEE ALSO: *Program Setup Dialog (General Property Sheet), Relocate Origin*

Drawing with Coordinate Systems

KeyCAD lets you change coordinate systems at any time while you are working. If you are drawing the outer wall of a house, for example, you may want to start the first wall at an absolute location in the drawing space, so you would use absolute coordinates for the first point. Each successive wall, however, would be defined by its length and angle relative to the first wall.

TIP: *Advanced users can "force" KeyCAD to use a coordinate system other than the current system while working in the Coordinate Fields. If you precede a coordinate with a $ sign, it will be interpreted as an absolute coordinate; if you precede it with an @ sign it will be interpreted as a relative coordinate.*

Command Reference

The following table summarizes the menu commands accessible from the Modes menu:

Command	Function
Snaps	Set the current snap mode.
Coord System	Set the current coordinate system.

Snap Tools

Menu: Modes|Snaps   **Toolbar:** Snap

The following table summarizes *KeyCAD*'s snap modes:

Command	Function
No Snap	Turn off all snap modes; point is defined at the location of the mouse click.
Snap to Vertex	Snap to the nearest end of a line segment, endpoint of an arc, or corner of a polygon or multiline.
Snap to Nearest on Graphic	Snap to the nearest point on any entity.
Snap to Midpoint	Snap to the midpoint of a line segment.
Snap to Arc Center	Snap to the center of an arc or circle.
Snap to Quadrant Point	Snap to the nearest quadrant point (at 0, 90, 180, or 270 degrees) on the circumference of a circle or arc.
Snap to Intersection	Snap to the nearest intersection of two line segments.
Snap to Grid	Snap to the nearest grid point or intersection of grid lines.
Ortho	Constrain the angles of lines; the default settings constrain them to 0, 90, 180, or 270 degrees. (You can change these settings in the Angle property sheet.)
Magnetic Point	Increase visual feedback; the rubber band line you are currently drawing will gravitate toward the snap point.

Combining Snap Modes

Some snap modes can be combined with one another, and some are mutually exclusive. No Snap turns off all other snap modes. Nearest on Graphic snap (which snaps to the nearest point on any object) also turns off all other snap modes. (Ortho and Magnetic point can stay on no matter which snap modes are active.)

The remaining snap modes all snap to a particular node (nodes are key points that define the object geometrically), and they are not mutually exclusive. You can, for example, have Snap to Grid and Snap to Vertex on simultaneously. If you then click the mouse while a grid point and a line segment are both within the snap aperture, your point will snap to

either an endpoint of the line segment, or to the grid point, whichever is closest to the mouse cursor.

NOTE: *The snap modes that can be used at the same time are all in the same section of the Modes|Snaps submenu.*

No Snap

Menu: Modes|Snaps|No Snap   **Toolbar:** Snap

Activating No Snap will turn off all snap modes except Ortho. With snap modes turned off, you define points by simply clicking at the point that you want to define in the drawing area, or by entering the point's coordinates on the status bar.

NOTE: *If Ortho mode is on, the second point of a line will snap to a point to constrain the angle of the line, even if No Snap mode is active.*

SEE ALSO: *Ortho*

Snap to Vertex

Menu: Modes|Snaps|Vertex **Toolbar:** Snap

When Snap to Vertex mode is active, the point you are defining will snap to the nearest vertex on an entity within the snap aperture. A vertex in *KeyCAD* can be:

•   The endpoint of any line segment (either a simple line or a segment of a multiline).

•   A corner of a polygon.

•   The endpoint of an arc.

NOTE: *When snapping to a double line, your point will be defined at the endpoint of either of the single line segments that comprise the double line, not to the double line's reference point.*
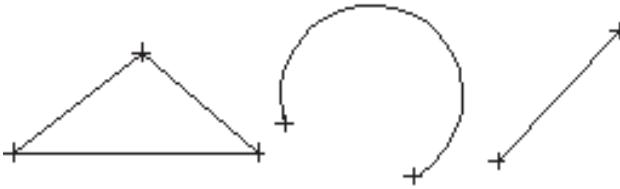
Types of vertices

Snap to Nearest on Graphic

Menu: Modes|Snaps|Nearest On Graphic   **Toolbar:** Snap

When Snap to Nearest on Graphic mode is active, a point will be defined at the nearest point on an object within the snap aperture.

Snap to Midpoint

Menu: Modes|Snaps|Middle Point   **Toolbar:** Snap

When Snap to Midpoint mode is active, the point you are defining will snap to the midpoint of a line segment. A line segment can be any part of an entity, such as one of the lines in a double line, or a segment of a multiline or polygon.
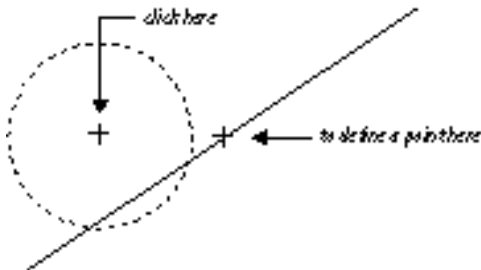
Snapping to the midpoint of a line segments

Snap to Arc Center

Menu: Modes|Snaps|Arc Center   **Toolbar:** Snap

When Snap to Arc Center is active, the point you are defining will snap to the center of an arc or circle that you indicate. To indicate the arc or circle, click on or near its perimeter.
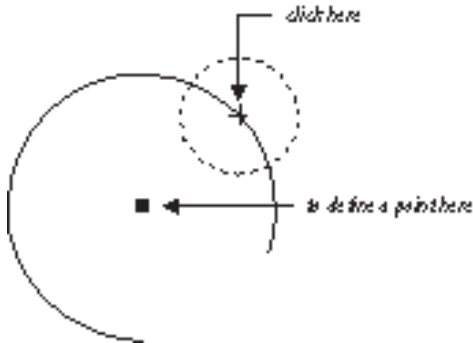
Snapping to an arc center

Snap to Quadrant Point

Menu: Modes|Snaps|Quadrant Point   **Toolbar:** Snap

When Snap to Quadrant Point is active, the point you are defining will snap to the nearest quadrant point. Quadrant points are at 0 degrees, 90 degrees, 180 degrees, and 270 degrees on the circumference of an arc or circle.
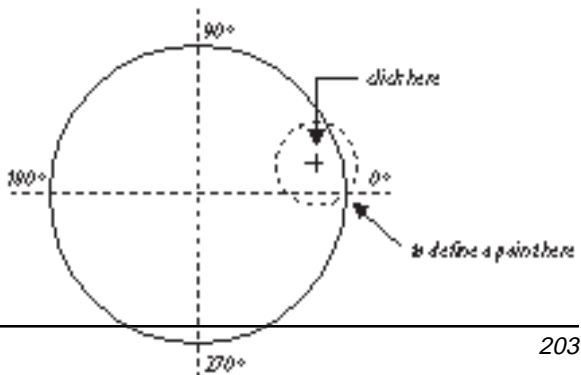
Snap to Intersection

Menu: Modes|Snaps|Intersection   **Toolbar:** Snap

When Snap to Intersection is active, the point you are defining snaps to the intersection of two lines. Both lines must be located within the snap aperture; but the intersection need not be located within the snap aperture.
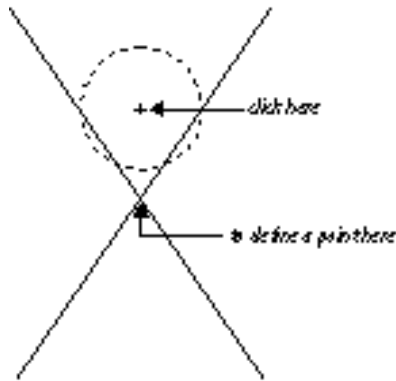
Snapping to the intersection of two lines

Snap to Grid

Menu: Modes|Snaps|Grid   **Toolbar:** Snap

When Snap to Grid is active, the point you are defining snaps to the nearest grid point, or to the nearest intersection of grid lines, if the grid is being displayed in lines rather than points or crosses.
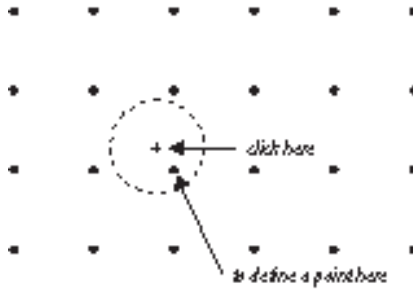
Snapping to points on the grid

SEE ALSO: *Grid Property Sheet*

Ortho

Menu: Modes|Snaps|Ortho **Toolbar:** Snap

Ortho mode constrains the angle of the entity you are drawing. By default, Ortho mode will constrain the angle of the entity so that it will be parallel to either the X or Y axis of the drawing. You can change these default settings in the Angle property sheet.
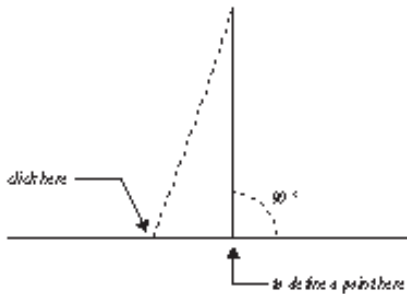
Drawing a line with Ortho mode active

SEE ALSO: *Angle Property Sheet*

Show Magnetic Point

Menu: Modes|Snaps|Show Magnetic Point   **Toolbar:** Snap

Magnetic Point provides visual feedback about the location of snap points. When this mode is active, as you move the mouse cursor within a critical distance from the snap point, the rubber band line will be attracted to the snap point, showing you its exact location. Unless speed is a consideration on your computer, or unless you are working on a very large drawing, it is probably useful to have this mode on all the time.
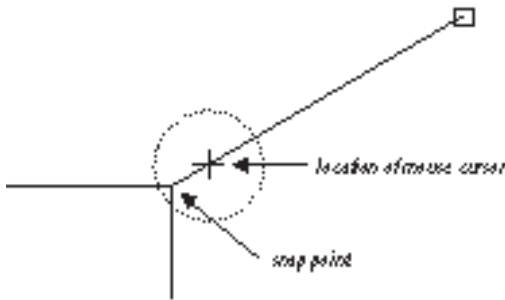
Rubber band line attracted to a snap point

SEE ALSO: *Snap Aperture*

Coordinate System

Menu: Modes|Coord System**Toolbar:** Snaps

The coordinate system menu commands enable you to change between absolute and relative, and to relocate the origin of the relative coordinate systems. The following table summarizes the coordinate system commands:

Command	Function
Absolute	Change to absolute Cartesian coordinates, in which you define points by their distance from a

fixed (absolute) point of origin.

Relative	Change to the relative coordinate system, in which you define points by their X and Y distance from the Relative Origin point.
Relocate Origin	Change the position of the Relative Origin point.

SEE ALSO: *Coordinate Systems*

Absolute Coordinates

Menu: Modes|Coord System|Absolute Coords   **Toolbar:** Snap

Use the absolute coordinate system to work in absolute Cartesian coordinates. When working in absolute coordinates, you define points by their X and Y distances from the Absolute Origin. This fixed point, absolute (0,0) is located by default at the lower-left corner of the drawing page (although the page can be moved using the Page Setup dialog).

For example, to draw a horizontal line two units long starting at (2,2), you would define the line's second point at (4,2).

SEE ALSO: *Coordinate Systems*

Relative Coordinates

Menu: Modes|Coord System|Relative Coords   **Toolbar:** Snap

Use the relative coordinate system to work in relative Cartesian coordinates. When you work in relative coordinates, you define the location of points relative to the Relative Origin (relative (0,0)).   To define a point relative to the current position, you can enter X and Y displacement values in the Coordinate Fields. You can enter either positive or negative values. The displacement values specify the distance of the new point from the Relative Origin.

NOTE: *The Relative Origin moves whenever you define a new point. In most cases, the Relative Origin will be the last point you defined. If you are drawing a circle or arc, however, the Relative Origin will be at the center of the circle or arc.*

TIP: You can fix the Relative Origin so that it does not move by checking the Fixed Relative Origin box in the Program Setup dialog (General property sheet). You can then move the Relative Origin manually using the Relocate Origin command.

SEE ALSO: *Coordinate Systems, Relocate Origin, Program Setup*

Relocate Origin

Menu: Modes|Coord System|Relocate Origin   **Toolbar:** Snap

Use the Relocate Origin command to manually relocate the Relative Origin point. When you start a new drawing, this point is located at the lower-left corner of the page, and appears as a red dot. You can relocate the point using any coordinate system; that is, you can specify its position in absolute or relative coordinates. To relocate the point of origin:

1. Choose Relocate Origin from the menu.

2. Define a point for the new origin by clicking the mouse in the drawing space or entering values in the status bar (the type of values you enter in the status bar depend on the currently active coordinate system).

SEE ALSO: *Coordinate Systems*

10
Setting Options
Tools Menu and Dialogs

The Tools menu gives you access to dialog boxes that let you control *KeyCAD* program and drawing settings. These features let you:

- Set program options that are saved with the application.
- Set drawing options that are saved with the drawing or template.
- Run *KeyCAD* scripts.

This chapter begins by discussing how to use some of the more powerful and useful features contained in these dialogs, specifically:

- Program and Drawing Setup
- Setting up Units and Scale
- Setting up the Grid
- Setting up Layers

The remainder of the chapter provides a complete reference to the dialogs and their controls.

Program and Drawing Setup

The Tools menu contains two large dialogs, Program Setup and Drawing Setup. Changes that you make in the Program Setup are saved whenever you exit *KeyCAD*: The next time you start *KeyCAD*, the settings will be exactly the ones you selected. Changes that you make in the Drawing Setup dialog are saved with your drawing, and they will come back exactly as you left them the next time you open the drawing in which they were saved.

If you have a standard way of setting up a drawing and you want to use the settings in other drawings as well, you can save the Drawing Setup options in a *template* using the Edit|Save As dialog. You can make the template that you use most frequently the *default template* (the template

that *KeyCAD* chooses if you don't specify a template) in the Create from Template dialog. You can also create multiple templates for different types of CAD tasks.

TIP: *To save the setup of your Desktop controls (that is, the display and position of toolbars and other interface controls), you must check the Save Desktop on Exit box in the General property sheet of the Program Setup dialog.*

SEE ALSO: *Create from Template Dialog, Drawing Setup Dialog, Program Setup Dialog, Save As*

Units and Scale

KeyCAD has two distinct "spaces": world space and paper space. The units in world space apply to the real-world object that your drawing models. The units in paper space apply to the paper on which your drawing will be printed. *Scale* is the ratio of Paper units to World units. A scale of 3/16" = 1', for example, means that an object 3/16" long in your drawing is modelling a real-world object that is one foot long.

Normally the coordinate system, the ruler, and the grid all show units in world space. You can switch to paper space by clicking the icon in the upper-left corner of the Ruler.

NOTE: *Dimensions always show World units, unless you manually override the dimension text.*

The Units and Scale property sheet and the Advanced Units property sheet give you control over:

- The type of units used in both paper space and world space.
- The scale of your drawing.
- The format and precision of the numbers in your drawing.

You can set these options either by choosing from one of *KeyCAD's* standard settings, which correspond to settings common to specific CAD disciplines, or you can specify custom settings.

SEE ALSO: *Advanced Units Property Sheet, Units and Scale Property Sheet*

Setting Up the Grid

The grid is a matrix, displayed as a pattern of dots, crosses, or lines that divides your drawing geometrically into Cartesian coordinate locations. You can make the grid visible or invisible using the View|Grid command.

You make the grid active or inactive by switching the Snap to Grid option on and off (using the Modes|Snaps|Snap to Grid menu option, the Snap toolbar, or the local menu). Note that you can snap to an invisible grid, and you can view an inactive grid: Whether the grid is visible has nothing to do with whether it is active.

KeyCAD has a *major grid* and a *minor grid*. The major grid shows larger divisions of your drawing space; the minor grid shows finer divisions of the drawing space between the major divisions. If you want to customize the grid setup, you can specify how many minor grid points there are between major grid points; and you can specify how frequently the grid points are visible, so that only larger grid divisions are visible, but you can still snap to the finer divisions.

You can also rotate the grid at an angle, or offset the grid so that it aligns with an object in the drawing.

For details on how to customize the grid, read the section on the Advanced Grid property sheet.

SEE ALSO: *Advanced Grid Property Sheet, Grid Property Sheet, Snap to Grid, View Grid*

Setting Up Layers

One of the most powerful ways you can organize a complex drawing is to create layers on which to place different categories of objects. Layers are like acetate sheets used in traditional drafting.

Layers in *KeyCAD* can be visible or invisible, editable or not editable. These properties allow you to view only the drawing objects that are relevant to the current task, and to protect objects on the drawing from inadvertent change.

You can also distinguish layers of the drawing by giving them distinct colors and line styles. You can assign colors and styles to layers in the Layers property sheet. Objects on the layer will then have the color and line style assigned to the layer, provided the relevant properties of the object are set to By Layer. For example, if you want an object's pen color to have the default color of the object's layer:

1.   Double-click on the object with the Select tool to go to the object's property sheets.

2.   Bring the Pen property sheet to the front.

3.   Set the Color property to By Layer.

4.   Click <OK> to close the dialog.

SEE ALSO: *Layers Property Sheet*

Scripts

KeyCAD 3.0 is a fully programmable CAD system. Its built-in BASIC interpreter allows programmers to create scripts in BASIC that can access any of *KeyCAD*'s functionality via the TCAPI or TCAPI32 DLL. You can create scripts using any text editor or word processor, then run the script directly from *KeyCAD* using the Tools|Run Script command. You can attach scripts to your drawing using the Scripts property sheet. Once scripts are attached to your drawing they appear in a submenu off of the Run Scripts menu item on the Tools menu.

Advanced programmers can create their own DLLs in any language that call *KeyCAD*'s exported functions.

For detailed information on scripting, *KeyCAD*'s BASIC language, or on the TCAPI functions, see the online help system.

SEE ALSO: *Run Scripts, Scripts Property Sheet*

Command Reference

The following table summarizes the commands accessible from the Tools menu. Only the major menu items are shown. The indented items on the menu are shortcuts that go directly to individual property sheets in the Program Setup and Drawing Setup dialogs. These property sheets will be described in detail in the following sections.

Command	Function
Program Setup	Display the Program Setup dialog, in which you can set options that are saved when you exit *KeyCAD*.
Drawing Setup	Display the Drawing Setup dialog, in which you can set options that are saved with your drawing.
Run Scripts	Run a *KeyCAD* script.

Program Setup Dialog

Menu: Tools|Program Setup

The Program Setup dialog contains five property sheets for controlling *KeyCAD* program settings, summarized in the following table:

Property Sheet	Functions
General	Specify template, read-only file access, user information, snap aperture, zoom factor, save, and advanced settings.
Desktop	Display and hide user-interface controls, control desktop colors and paper display.
Groups	Set options for naming groups.
File Location	Specify directories for autosave, backup, template, and script files.
Color Palette	Add new colors to the palette, modify existing colors, and delete colors from the palette.

You can access each of these property sheets directly from the Tools menu, which gives you the convenience of opening the Program Setup dialog with the correct property sheet in front using a single command. The options that you set in this dialog are saved when you exit *KeyCAD*, and will remain in effect the next time you start the program.

General Property Sheet of Program Setup Dialog

Menu: Tools|Program Setup

In the General property sheet, you can set general program options and user information. The controls on this property sheet enable you to:

- Set the default template used when you start a new drawing, and specify whether *KeyCAD* will prompt you for a template when you start a new drawing.
- Specify whether drawings are opened as read-only files.
- Read and update user information.
- Set the zoom factor and snap aperture.
- Choose Geometric Selection or Fixed Relative Origin.

General property sheet of Program Setup dialog

The General property sheet contains the following controls:

Default Template for New Files. If you do not specify a template when you start a new drawing, *KeyCAD* will use a default template. To change the default template, click the arrow on the right side of the drop-down list and choose a template (or type a template name directly into the box).

Prompt for Template Name. Check this box if you want *KeyCAD* to prompt you for a template whenever you start a new drawing. If you want to use the default template without being prompted, clear this check box.

SEE ALSO: *Save As, Open, New*

Open All Files as Read-Only. There may be times when you want to look at drawing files without changing them, or prevent them from being inadvertently changed by someone else. Check this box to prevent anyone using *KeyCAD* from changing existing files. To restore the ability to change files, clear the check box.

User Information. The User Information text fields contain the user and company name.

Snap Aperture. The *snap aperture* is a circular area around the mouse cursor. *KeyCAD* will detect and snap to a potential snap point only if it is within the snap aperture. (Which points qualify as potential snap points depends on the current snap mode.) You can set the radius of the snap aperture using the snap aperture control.

Zoom Factor. The zoom factor setting controls how much you zoom in or out of the drawing when you use the Zoom In (<Gray +>) or Zoom Out (<Gray ->) command. The default zoom step setting is 2, which is a factor, meaning that if you zoom in one step, objects will be twice as big; if you zoom out one step, objects will be half as big, and the area of the drawing shown in the view will double. You can set the zoom step to any number between 1.01 and 2000.

TIP: *You can view the Snap Aperture as you work by choosing View|Aperture.*

SEE ALSO: *Snap Aperture, Snap Modes, Snap Tools, View Crosshairs, Zoom In, Zoom Out, Zooming*

Save Options

The General property sheet includes several controls that allow you to control aspects of how *KeyCAD* saves drawing files.

Prompt for summary information. If this box is checked, *KeyCAD* will display the Summary Information dialog, in which you can record information about your drawing, the first time that you save any drawing to disk, and whenever you use Save As to save a new copy of the file.

Make backup copies. If this box is checked, *KeyCAD* will automatically make backup copies of your drawings whenever you save them. You can choose either the On First Save or On Every Save option button to determine how often *KeyCAD* saves your backup files. Backup files share the same file name as the drawing file, but have the file extension .BAK instead of .KCW. By default, backup files are saved in the same directory as your primary drawing files. You can specify another directory in the File Location property sheet of this dialog.

Save drawings every. If this box is checked, *KeyCAD* will make autosave copies (.ASV files) of your document automatically at regular intervals. You can set the duration of the intervals in minutes using the Minutes control just below the check box.

Save desktop on exit. If this box is checked, *KeyCAD* will save the desktop settings when you exit the program. This means that *KeyCAD* will "remember" which toolbars and other controls you have displayed, as well as their positions on the desktop, and will display them exactly the same way the next time you start the program.

SEE ALSO: *Save, Save All, Save As, Desktop Property Sheet*

Advanced Settings

Fixed Relative Origin. Check this box if you want to fix the Relative Origin in place while you are in relative coordinates. You will have to relocate the Relative Origin manually using the Modes|Relocate Origin command.

Geometric Select Edit. Check this box if you want to select objects by their geometric extents, rather than their cosmetic extents. This permits geometric accurate scaling of objects like double lines and lines that have width, because the geometric and cosmetic extents differ.
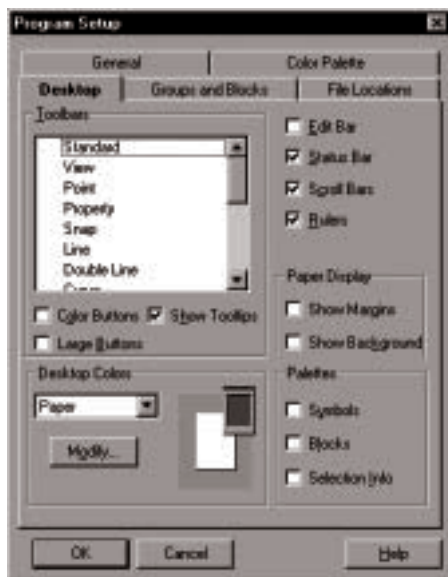
SEE ALSO: *Coordinate Systems, Geometric and Cosmetic Select Modes*

Desktop Property Sheet

Menu: Tools|Desktop

The Desktop property sheet contains controls that let you:

- Display and hide toolbars and other interface controls.
- Display and hide the paper margins and background.
- Set the color of the paper and background.
- Set the zoom factor.

Desktop property sheet

NOTE: *Many of the functions available on the Views property sheet are also available from the View menu.*

SEE ALSO: *View Toolbars Dialog, View Grid, View Crosshairs, View Rulers, View Edit Bar, View Symbol Library, View Metrics, Zoom In, Zoom Out, Zooming*

Toolbar Controls

Toolbars. The Toolbars scrolling list displays the names of all the toolbars available in *KeyCAD*. Toolbars that are currently displayed are checked. To display a hidden toolbar, scroll through the list and click on the check box next to the name of the toolbar, displaying the check. To hide a currently displayed toolbar, click on the box, removing the check.

Color Buttons. Check this box for color buttons, clear it for grayscale buttons.

Large Buttons. Check this box for large buttons, clear it for standard size buttons.

Show Tooltips. Check this box to display tooltips (the yellow boxes that appear under the button if you hold the mouse cursor over them); clear the box to suppress display of tooltips.

NOTE: *This function is also available in the View Toolbars dialog.*

Desktop Controls

The Views property sheet contains a number of check boxes that control the display of a variety of user interface controls and palettes, specifically:

- Edit Bar
- Status Bar
- Scroll Bars
- Rulers

Checking a box displays the corresponding control; clearing the box hides the control.

TIP: *It's a good idea to switch off controls that you don't use, because this increases the size of your drawing area.*

Paper Margins and Background

The Paper Display control group lets you display or hide the paper margins, and set whether the background behind the paper is colored or white:

Margins. Check this box to display page margins; clear the box to hide them.

Background. Check this box to display a background behind the paper; clear the Background check box to display a white background.

TIP: *You can switch off the paper until you're ready to print, then use the Fit feature of the Page Setup Dialog to place your drawing on the page.*

Desktop Colors

The Desktop Color control group contain buttons for customizing the color of the paper, background, and floating palettes. Click in the drop down box and Select Paper, Background, or Floater. Click the Modify button to change the color using the Color Picker.

SEE ALSO: *Using the Color Picker*

Groups Property Sheet

Menu: Tools|Groups

The Groups property sheet lets you control whether and how *KeyCAD* generates names for groups, and set whether *KeyCAD* prompts for group names.

Groups property sheet

TIP: *The name of a group is displayed on the General property sheet of the Properties dialog. To view this dialog, you can simply double-click on a group.*

SEE ALSO: *Create Group, Properties Dialog*

Groups Options

The following controls let you determine how *KeyCAD* handles the naming of groups:

Generate group names. If this option is checked, *KeyCAD* automatically generates a name whenever you create a group.

Group name prefix. If the Generate Group Names option is on, you can type a string (a series of characters) into the Group Name Prefix text box, and *KeyCAD* will use this string to automatically generate group names. All the characters that you type into the box are used literally, except the '@' character, which is a placeholder for a group number that *KeyCAD* generates automatically. You can type the '@' character in the name at the place where you want the automatic number to appear. For an explanation of how this works, see "Automatic Group Names," which follows the next section.

Prompt for name. Check this option if you want *KeyCAD* to prompt you for a name every time you create a new group.

NOTE: *If you turn off both the Prompt for Name and Generate Group Names options, KeyCAD will not name groups.*

TIP: *You can subsequently name or rename a group by editing its properties. To do this:*

1. *Select the group that you want to rename.*
2. *Double-click on the group to bring up the Properties dialog.*
3. *Type a name for the group into the Attribute text field, then press <Enter> to close the dialog.*

SEE ALSO: *Create Group, Explode, Properties Dialog*

Automatic Group Numbering

When *KeyCAD* creates a group name, it automatically gives the group a number, which increases by 1 whenever you add a group to the drawing. This ensures that all groups have unique names. By default, the names that *KeyCAD* generates are "Group 1", "Group 2", and so forth.

You can change the default that *KeyCAD* uses for automatically naming groups in the Group Name Prefix text box. The names that you type in these text boxes have two parts: the name itself (any series of characters), and the '@' character, which is a placeholder for the automatic number. You can place the '@' character anywhere in the name, to specify where *KeyCAD* will place the automatic number.

For example, if you type "Part @" in the Group Name Prefix text box, then create the first group in a drawing, the group will automatically be named "Part 1", and the next group you create will be named "Part 2". If you type "#@ gear" into the Group Name Prefix text box, then create the first group in a drawing, the group will be named "#1 gear", and the next group you create will be named "#2 gear".

NOTE: *The first '@' that you include in the name is a placeholder for the automatic number. Any '@' characters that occur after the first one will be part of the literal name.*

SEE ALSO: *Create Group, Group Options*

File Location Property Sheet

Menu: Tools|File Location

File Location property sheet

The File Location property sheet allows you to specify the directories where *KeyCAD* places these different types of program files:

Autosave (.ASV). These are program files that *KeyCAD* saves automatically, if this option is turned on in the Save property sheet. These will allow you to restore work that would otherwise be lost due to a system crash.

Backup (.BAK). These are copies of drawing files that *KeyCAD* saves, if you check the Make Backup Copies option in the Save property sheet.

Scripts (.BAS). Files containing Enable BASIC scripts. This setting specifies the default directory after you choose Tools|Run Script.

Templates (.KCT). Files in which you save settings, as well as symbol libraries, and scripts, that you can start with as the basis of a new drawing.

To specify a directory in which to place a type of file:

1. In the list box, select the file type for which you want to specify a directory.

2. Click the modify button. *KeyCAD* will display a standard Windows Open dialog.

3. Use the standard Windows dialog to navigate to the drive and directory where you want to place the files. Click Open (or OK if you are using Windows 3.1) to accept the directory.

TIPS: *If you have two disk drives (or access to a network drive), it is wise to have your backup files automatically saved to a second drive. If the drive that contains your drawing files is damaged, you can recover your backup files from the second drive.*

Color Palette Property Sheet

Menu: Tools|Color Palette

The Color Palette dialog displays all the colors that are currently available. These colors and their names are displayed in a scrolling list, and also in a palette control to the right of the scrolling list. The Color Palette dialog lets you add new colors to the palette, modify existing colors, and delete colors from the list.
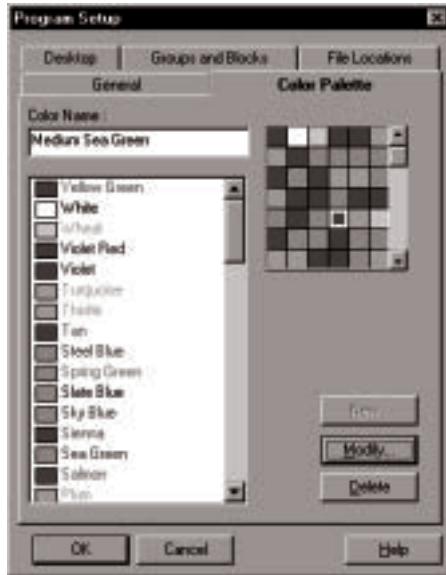
Color Palette property sheet

Adding a New Color

To add a new color:

1. Type a name for the color in the Color text box.

2. Click the New button, which displays the Color Picker dialog.

3. Use the Color Picker dialog to define a new color, then click the OK button.

The new color that you defined will appear in the scrolling list and palette controls. This color will now be available for formatting the pen and brush colors of objects.

SEE ALSO: *Using the Color Picker*

Modifying an Existing Color

To modify an existing color:

1. Select the color that you want to modify in the scrolling list or in the palette control.

2. Click the Modify button, which displays the Color Picker dialog.

3. Use the Color Picker dialog to redefine the color, then click the OK button.

Notice that the color is now changed in the scrolling list and palette controls. The modified color will now be available for formatting the pen and brush colors of objects.

SEE ALSO: *Using the Color Picker*

Deleting a Color

To delete a color that you no longer need to use in a drawing:

1. Select the color in the scrolling list or in the palette control.
2. Click the Delete button.

NOTE: *Objects that have already been colored in a color that you delete will still have the deleted color.*

Using the Color Picker

Color Picker dialog

The color picker lets you specify colors that you can add to your drawing's color palette, or use to replace an existing color in the palette. The dialog contains a color wheel, which displays a spectrum of available colors, and a scrollbar with which you can control the overall brightness of the colors on the wheel. It also contains controls that let you define a color by directly entering its RGB (Red, Green, Blue) values, or by specifying its Hue, Saturation, and Value, which correspond to positions on the color wheel and scrollbar.

A lower color display box to the left of the color wheel gives you visual feedback about the color you have chosen. If you are modifying an existing color, the original color will be displayed in the upper color display box.

Picking Colors with the Mouse

To pick a color with the mouse:

1. Set the overall brightness of the color wheel using the scroll bar to the right of the wheel.
2. Click at the position on the wheel that is the color and shade you want.

An example of the color you have picked will appear in the color display box to the left of the color wheel.

Setting RGB Values

Screen colors can be defined by a combination of Red, Green, and Blue (RGB) values. Each of these values can be set to an integer from 0 to 255. For example, an RGB setting of (0,0,0) is black; an RGB setting of (255,255,255) is white. A setting of (255,0,0) is "pure" red, and a setting of (0,255,255) is Cyan.

To define a color by its RGB values, enter the values in the R, G, and B controls on the dialog. Notice that the color in the lower color display box changes to correspond to the values you have entered, and the position of the color picker on the color wheel changes as well.

Setting Hue, Saturation, and Value

Hue and Saturation settings correspond to physical positions on the color wheel. Hue represents an angular position on the wheel measure in degrees (corresponding to the color spectrum), and can have a value from 0 to 360. Saturation represents a radial position on the wheel (corresponding to the amount of "grayness" in the color), and can have values from 0 to 255. Value controls the overall brightness on the color wheel, so changing the value in this control is the same as using the scroll bar to the right of the color wheel.

Drawing Setup Dialog

Menu: Tools|Drawing Setup

The Drawing Setup dialog includes five property sheets containing settings that are saved in a drawing or template file. These property sheets are summarized in the following table:

| Property Sheet	Functions
Display	Set options for optimizing redraw speed.
Grid	Choose the grid type and other grid options.
Symbol Libraries	View and maintain symbol libraries in current drawing or template.
Units and Scale	Set the unit of measure, the drawing scale, and related options.
Angle	Choose direction, units, and base angle for angular measurements.
Scripts	View and maintain scripts in current drawing or template.
Layers	Create new layers, modify existing layers, and delete layers.
Advanced Grid	Control the major and minor grid and grid origin.
Advanced Units	Set paper, text, and line width units.

The following sections describe these property sheets and their controls.

Display Property Sheet

Menu: Tools|Display

The Display property sheet contain options for adjusting the quality and redraw speed of the screen display.
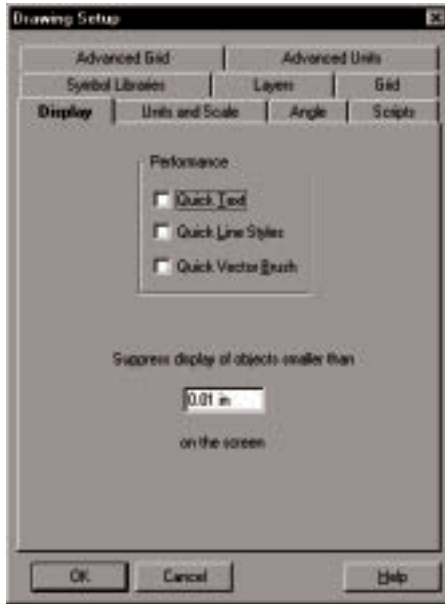
Display property sheet

Quick text. Check this box to display all text as small boxes.

Quick line styles. Check this box to simplify the display of line styles.

Quick vector brush. Check this box to simplify the display of hatch patterns.

Suppress display of objects smaller than... Use this spin control to set the size at which *KeyCAD* will draw a simplified representation of objects. This size is measured in device units (the "device" being your computer's display), so that one inch will be equal to approximately one inch of space on your screen. At smaller sizes, *KeyCAD* will display entities using a simplified representation, decreasing the amount of time that *KeyCAD* needs to calculate and display the entity.

TIP: *If you are working in a very large drawing or if you are using a relatively slow system, you may be able to increase your screen redraw speed by clicking one or more of the options in the Performance control group of this property sheet. The amount of speed that you gain depends on the amount of text, styled lines, and fill patterns that your drawing contains.*

Grid Property Sheet

Menu: Tools|Grid

The grid is a matrix, displayed as a pattern of dots, crosses, or lines, that divides your drawing geometrically into Cartesian coordinate locations.

The Grid property sheet allows you to control the type of the grid type and display style, how far apart the grid points are spaced, and the color of the grid lines.
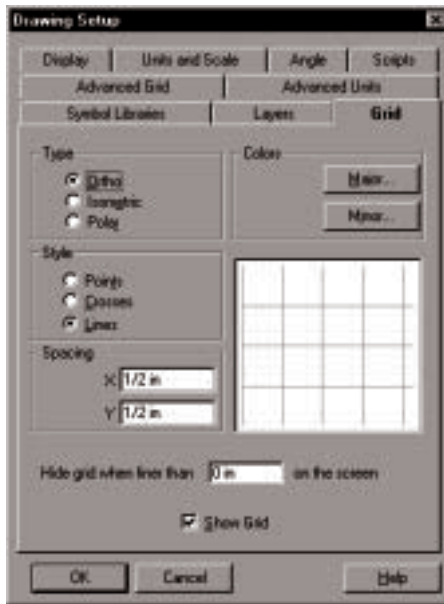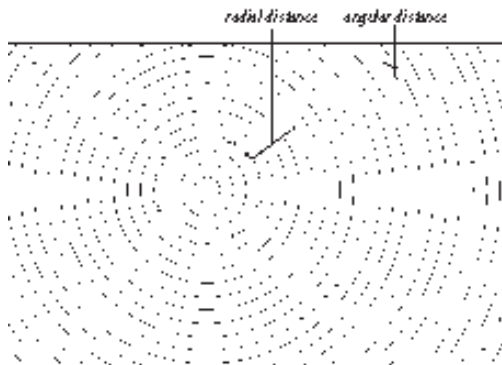

Grid property sheet

The following sections describe the controls of the Grid property sheet.

Grid Type

Click the appropriate option buttons in the Type area of the property sheet to set the grid to any one of the following types:

Ortho. Choose this option for an orthogonal grid. Points of the orthogonal grid are aligned parallel to the X and Y axes of the Cartesian plane.

Isometric. Choose this option for an isometric grid. Points in the isometric grid are aligned along lines measured 30º, 90º, and 150º from the horizontal axis. This type of grid is used in isometric drawing, which uses perspective drawing techniques to represent three-dimensional objects.
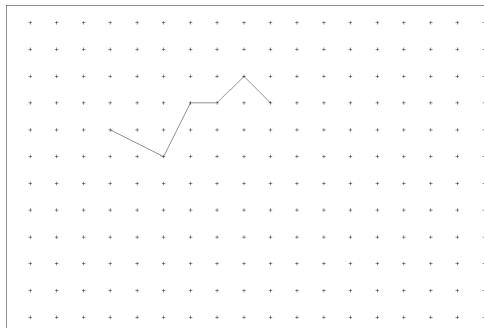
Orthogonal and isometric grids

TIP: *Advanced users can set the base angle of the grid to something other than horizontal, using the Base setting in the Advanced Grid property sheet.*

SEE ALSO: *Coordinate Systems*

Grid Color

Click the buttons in the Color group to choose a new color for the major and minor grid. Click the Major or Minor button, then choose a color using the Color Picker.

SEE ALSO: *Using the Color Picker*

Grid Style

Choose from the following Grid Style options:

Lines. Click this option to display the grid as intersecting lines, similar to graph paper.

Crosses. Click this option to display the grid as a matrix of crosses.

Points. Click this option to display the grid as a matrix of points.

Grid Spacing

The *spacing* of the grid determines how far apart the grids or lines of the grid are spaced from one another. For example, if spacing is set to one foot, points on the grid will be spaced a foot apart. Grid spacing is set in either World or Paper units, whichever is currently being used.

If you have chosen an orthogonal grid in the Grid Type control:

1. Type the amount of horizontal space between each grid point in the X box.
2. Type the amount of vertical space between each grid point in the Y box.

NOTE: *Grid spacing applies to every grid point, regardless of whether it is a minor or major grid point, and without regard to the frequency of the grid.*

Other Grid Controls

The Grid property sheet also contains these controls:

*Hide grid when finer than...* You can type a value in this box to control how fine a grid can be drawn on the screen. When the grid becomes finer than this value, it will not be drawn. Note that this value applies to distances on your computer screen, which are not related to World or Paper units.

Show grid. Check this box to display the grid. This is the same as choosing View|Grid from the menu.

Advanced Grid Property Sheet

The Advanced Grid property sheet gives you precise control of:

- Grid *frequency,* which determines how visible grid lines alternate with invisible grid lines.
- The *minor grid*, or the finer grid points that appear between points on the major grid.
- The grid *origin*, which lets you control how the grid aligns itself to the coordinate system and to drawing objects.
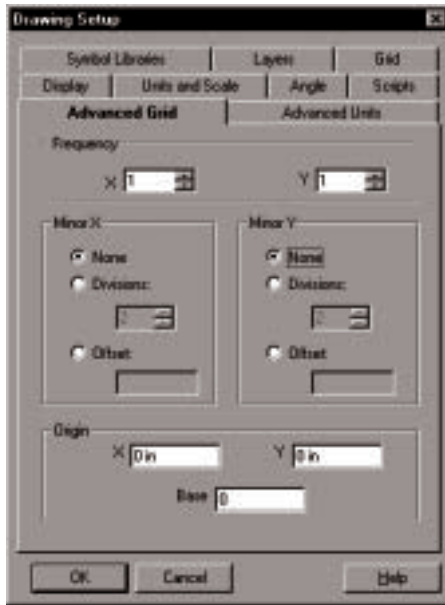
Advanced property sheet of Grid property sheet

Grid Frequency

The Frequency settings lets you alternate visible grid lines with invisible grid lines. For example, if you set the frequency 2, then every other grid line will be displayed. If you set frequency to 4, then every fourth grid line will be displayed. You will still be able to snap to the invisible grid lines.

⚠ WARNING: *Don't confuse this with whether the grid itself is visible. If you make the grid invisible using View/Grid, then the whole grid will be invisible.*

If you are customizing an orthogonal or isometric grid:

1. Type a value in the X box to set the frequency of horizontal grid points.
2. Type a value in the Y box to set the frequency of vertical grid points.

Minor Grid Settings

There are two control groups for minor grid settings: Minor Grid X, for setting horizontal minor grid points, and Minor Grid Y, for setting vertical minor grid points. You change the settings in these two control groups independently, using identical procedures. Each group contains three option buttons:

None. Select this option to turn the minor grid off. This means that all of the grid points or lines will be displayed as major grid points.

Division. Select this option to turn the minor grid on. You can set the number of divisions between each major grid point using the spin button control near the option button.

Offset. Select this option to create minor grid points at a fixed distance (offset) from each major grid point. You can use positive or negative numbers.
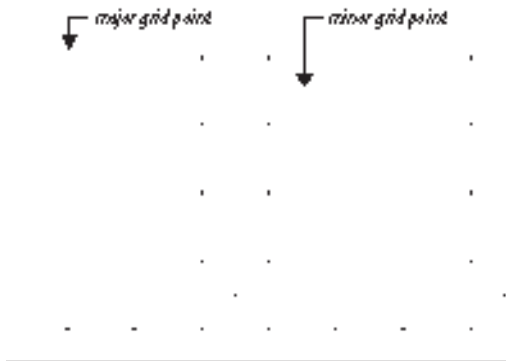

Major and minor orthogonal grid

Grid Origin

The grid origin is located by default at the origin of the absolute coordinate system, which is at coordinate (0,0). You can reset the origin point of the grid, causing the grid to be offset from the coordinate system

locations. You can also change the base angle of the grid, aligning it to an object that is not oriented orthogonally in the drawing space (for example, a house that is oriented at an angle in its lot).

To reset the grid origin, type absolute coordinate values into the X and Y text boxes. To change the base angle of the grid, enter the angle in the Base text box.
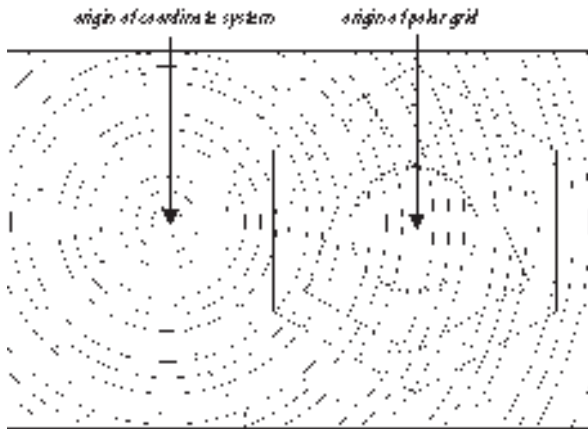
Offset grid origin

TIP: *This feature allows you to use a "local" grid. If you are working in an area of your drawing you can relocate the grid origin so that it aligns to the set of objects you are working on, rather than to the drawing space as a whole.*

Symbol Libraries Property Sheet

Menu: Tools|Symbol Library

KeyCAD 3.0 symbol libraries are actually just Windows folders that contain *KeyCAD* drawings. This dialog associates descriptive names with folders that you want to use as symbol libraries. Once you create names for the symbol libraries in your drawing, you can easily switch between the named symbol libraries using the Symbol Library palette. These libraries will be saved with your drawing, or in a template if you save your drawing as a template.

Symbol Libraries property sheet

SEE ALSO: *View Symbol Library*

Adding a New Symbol Library

The Add button lets you associate a descriptive name with a symbol library.

1. Type a name for the library in the Name text box.
2. Click the Add button. *KeyCAD* will display the standard Windows Open dialog. Use the dialog to choose the folder containing the library, then click OK to return to the Drawing Setup dialog.

When you select a library, its name, file name, and path appear in the property sheet's list box.

Deleting a Symbol Library Name

To delete a symbol library's descriptive name from the list, select its name in the list box, then click the Delete button.

NOTE: *Deleting a name removes the association from your drawing, but has no effect on the library's folder or the drawing file contained in the folder.*

Changing the Symbol Library Association

To associate a descriptive name with a different symbol library:

1. Select the symbol library name that you want to change by clicking on it in the Name list box.

2. Click the Modify button. *KeyCAD* will display the standard Open dialog. Use the dialog to navigate to the directory with which to associate the symbol library name, then click OK to return to the Drawing Options dialog.

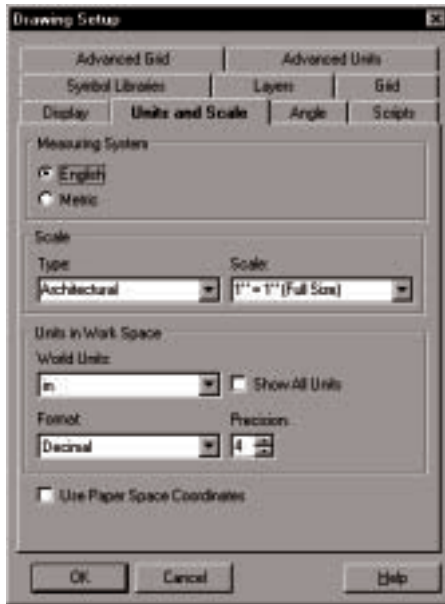# Units and Scale Property Sheet

Menu: Tools|Units and Size

The Units and Size property sheet contains options for setting the drawing scale and for controlling the display of linear units.

TIP: *KeyCAD's built-in templates and wizards can help you set up your drawing space when you start a new drawing. This property sheet is for cases when you want to set up your units and scale manually.*

Units and Scale property sheet

The following sections describe the controls available on this property sheet.

Measuring System

The Measuring System control group contains two general settings, English and Metric. Choosing either of these settings will select different sets of options to choose from in the other controls of this property sheet.

Scale

Scale is the ratio of measurements on the page (Paper units) to measurements in the object being modelled (World units). *KeyCAD* lets you set the drawing scale using one of three standard systems: Architectural, Metric, and Absolute. If Metric is selected in the Measuring System group, then Absolute will be the only choice available. The following table shows some sample drawing scales expressed in each of these systems:

System	Example
Architectural	3/16" = 1'-0"
Engineering	1"= 200'
Absolute	1:500

Use the Type drop down box to choose a system, then use the Scale box to choose a scale, or type in a custom scale.

Units in Work Space

Use the controls in this group to specify a unit of measure for your drawing:

World Units. Click in this drop-down list to choose a unit of measure. The choices that appear in the list will depend on whether English or Metric is selected in the Measuring System group.

Show All Units. Check this box if you want the World Units drop-down list to display all available units, regardless of whether English or Metric is selected in the Measuring System group.

Format. Click in this box to choose a format for the display of numbers. The choices available in this box will be limited to Decimal and Scientific if Metric is chosen in the Measuring System group. The following table shows examples of the available number formats:

Format	Example
Decimal	3.25
Scientific	1E+01 mm
Fractional	3 3/16
Architectural (Feet in Fractional)	1'-5 11/16"
Engineering (Feet in Decimal)	1"= 200'

Precision. Use this spin control to specify the precision of decimal numbers (the number of places to the right of the decimal point that *KeyCAD* displays). You can either click on the arrows or type a number directly into the box.

Use Paper Space Coordinates

Check the Use Paper Space Coordinates box if you want the Ruler, Grid, and the coordinate system to be in Paper, rather than World, units. This is useful if you want to precisely arrange your drawing on a page, or if you want to precisely place or scale an element of your drawing, such as text, in Paper space.

Tip: *Checking and unchecking this box is exactly the same as clicking the Paper/World button in the upper-left corner of the Ruler.*

Advanced Units Property Sheet

The Advanced Units property sheet contains controls for setting units in Paper space.

Advanced Units property sheet

The property sheet contains the following controls:

Paper Units. Click in this drop-down list and scroll to choose a unit for setting and displaying Paper measurements on the Ruler, in the coordinate system, on the grid, and in the Edit Bar.

Text Units. Click in this drop-down list and scroll to choose a unit for setting and displaying the size of text.

Line Width Units. Click in this drop-down list and scroll to choose a unit for setting and displaying the width of lines in the Pen property sheet of the Properties dialog.

Angle Property Sheet

Menu: Tools|Angle

The Angle property sheet gives you precise control over the measurement and display of angles, as well as control over the Ortho angle. You can change the system and direction of measurement, the base angle, and the precision of measurement.
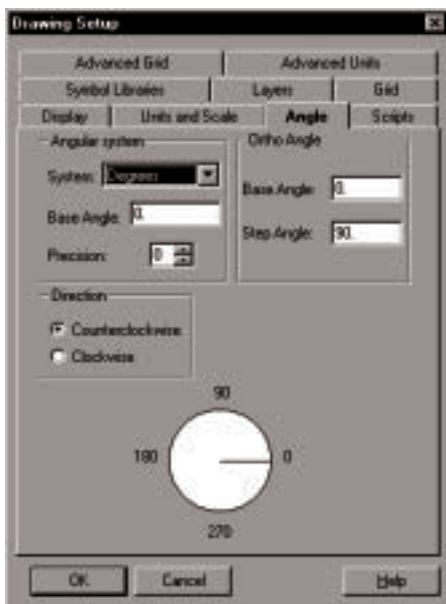
Angle property sheet

Angular System

Use the list box to choose a system appropriate to your drawing. You can choose to display angular measurements in degrees, degree minutes and seconds, grads, radians, or the surveyor system. In addition, you have the following two options:

Base Angle. Type a value into this text box to set the position of the base angle. Setting the base angle to 90 degrees, for example, would cause *KeyCAD* to start measuring angles from the top quadrant point of a circle or arc, rather than from the right quadrant point.

Precision. Use this control to set the precision of the display of decimal numbers. If you set Precision to 2, for example, measurements will be displayed to an accuracy of two digits to the right of the decimal point.

Direction

Use these option buttons to choose whether to measure angular distances in a clockwise or counterclockwise direction.

Ortho Angle

Use these controls to customize the Ortho angle. Ortho constrains the angles of lines during drawing and editing operations.

Base Angle. Type a value into this text box to set the position of the ortho base angle, which is the angle from which the ortho angle is measured. Changing this angle can be helpful if you want the ortho angle to be measured in relation to an object that is not oriented orthogonally relative to the drawing space.

Step Angle. The step angle determines the angle to which Ortho constrains lines. If you set the step angle to 15, for example, the line will be constrained to angles 15 degrees apart. *KeyCAD*'s default sets the step angle to 90 degrees.

Scripts Property Sheet

Menu: Tools|Script

The Scripts property sheet associates descriptive names with BASIC script files, and allows users to browse descriptions embedded within script files. For specific information about the Enable Basic language, or about how to access *KeyCAD*'s internal functions, see the *KeyCAD* help system.

Scripts property sheet

Adding a New Script

The Add button lets you associate a descriptive name with a script file
and run the script using the descriptive name from the script picker.

1. Type a name for the script in the Name text box.
2. Click the Add button. *KeyCAD* will display the standard Open
   dialog. Use the dialog to choose the script's .BAS file, then click
   OK to return to the Drawing Setup dialog.

When you select a script, the name of the script, as well as its file name
and path, appear in the scripts list box. If descriptive information is
available in the script's file, this information will appear in the
Description area of the property sheet.

Once the script is added to your drawing, it will appear in a submenu to
the right of the Tools|Run Scripts menu. You can run the script
immediately by choosing it from the submenu.

Deleting a Script

To delete a script's descriptive name from the list, select its name in the scripts list box, then click the Delete button.

NOTE: *Deleting a script removes the script's association with the drawing, but leaves the script's .BAS file intact, so that you can add it to your drawing or template again at any time.*

Overwriting an Existing Script

To replace an existing script in a drawing or template:

1. Select the script that you want to replace by clicking on it in the scripts list box.
2. Click the Modify button. *KeyCAD* will display the standard Open dialog. Use the dialog to choose a .BAS file with which to replace the existing script, then click OK to return to the Drawing Option dialog.

Layers Property Sheet

Menu: Tools|Layer

The Layers property sheet allows you to create new layers, set the current drawing layer, set the properties of layers, and delete layers.

Layers property sheet

Layers are virtual levels in a drawing, analogous to acetate sheets (overlays) in traditional drafting. Layers give you the advantage of sorting objects in your drawing by the type of object they are modelling, by order of creation, or by any principal that suits the way you work. You can then place each category of object on a separate layer of the drawing, so that you can work on one layer of objects at a time.

Layers can be visible or not visible, editable or not editable. If you want to view only a subset of the objects in your drawing, set the layers you want to see as visible, and set all other layers to invisible. Setting layers to non-editable makes it possible to protect objects on those layers from unintended change while someone is working on another layer of the drawing.

Layers can have their own default color and line styles. If the color and line-style properties of objects on a layer are set to By Layer, they will have the color and line style assigned to the layer.

SEE ALSO: *Format Properties*

NOTE: *If you create a group composed of objects on different layers of the drawing, the group will reside on the current layer (the layer that is current at the time the group is created). If you subsequently explode the group, each of the constituent objects will be on its original layer.*

Creating a New Layer

To create a new layer, type a name for the layer in the Name text box, then click the New button. The new layer name will appear in the Layers dialog list box.

Setting the Current Drawing Layer

The current drawing layer is the layer on which any new objects that you create or insert will reside. To set the current drawing letter, select the name of the layer from the Layers dialog list box, then press <Enter>.

Setting Layer Properties

You can set the properties of a layer using the controls in the Properties area of the Layers dialog. Any changes you make apply to the layer that is currently selected in the Layers dialog list box.

Visible. Check this box to make the layer visible and uncheck it to make the layer invisible.

Editable. If this box is checked you can draw, insert, and edit objects on the layer. If it is not checked, then you can view the layer, but you can't change it.

Color. Use this pull-down box to select a color for the layer. If objects on the layer have their pen or brush color property set to By Layer, then the pen or brush of the objects will have the color specified here.

Line Style. Use this pull-down box to select a line style for the layer. If objects on the layer have their line style property set to By Layer, then the objects will have the line style specified here.

Order. Use this text box to assign an *order number* to the layer selected in the Layers dialog list box. Order numbers determine the order in which layers are drawn (which corresponds to the order in the Layers list box). Objects drawn on layers with higher order numbers will overlap objects drawn on layers with lower order numbers. The order numbers of layers are all set to 0 initially. Layers that have the same order number are sorted in alphanumeric order.

After you have made changes to the properties of the selected layer, click the OK button to apply the changes.

NOTE: *The Bring to Front and Send to Back commands (on the Format menu) affect the redraw order of objects within a single layer, or across layers if the layers have the same order number. An object on a layer with a higher order number will always overlap an object at the front of a layer with a lower order number.*

SEE ALSO: *Bring to Front, Format Properties, Send to Back*

Deleting a Layer

To delete a layer, you must first clear any objects on the layer. You can select the objects on a layer using the Edit|Select By|Layer dialog, then clear the selected objects. Once the layer is cleared of objects, you can delete it in the Layers dialog by selecting its name in the list box and clicking the Delete button.

SEE ALSO: *Select by Layer Dialog*

Run Scripts

Menu: Tools|Run Scripts   **Toolbar:** Script

The Run Scripts command lets you run scripts that you have placed in your drawing, or browse for scripts located anywhere on your system. Scripts are created in *KeyCAD*'s built-in BASIC language using Notepad or any text editor or word processor. You can place scripts in your drawing using the Scripts property sheet (Tools|Script).

Scripts that you have placed in your drawing will appear in a submenu off the Run Script menu, followed by a More Scripts menu item. The scripts will also appear as buttons on the Script toolbar. You can run one of these scripts by choosing it from the submenu or clicking its tool button.

If you want to run a script that is not placed in your drawing, just choose Run Script (if no scripts are placed in your drawing), or More Scripts from the submenu (if there are scripts placed in your drawing). You can also click the Run Scripts tool shown in the heading of this section. The Run Script dialog will appear.

Run Script dialog

Use this standard Windows dialog to browse to the script that you want to run, the press <Enter> to run the script. For additional information on creating scripts, see the online help system.

NOTE: *KeyCAD's Scripts folder contains some sample scripts that you can run and explore.*

SEE ALSO: *Scripts Property Sheet*

11

Working with Multiple Documents

Windows Menu

KeyCAD uses a *multiple document interface*. This means that the *KeyCAD* application window can contain multiple document windows, each of which contains a *KeyCAD* drawing. Whenever you open a *KeyCAD* drawing or start a new *KeyCAD* drawing, the drawing will appear in its own window, within the *KeyCAD* application window.

If you are familiar with other Microsoft Windows programs, you already know how to work with windows and minimized window icons. For instruction on basic techniques, such as moving, sizing, restoring, maximizing, and minimizing document windows under Windows 3.1 and Windows 95, see your Microsoft Windows documentation. These topics will not be covered in this chapter.

One big advantage of the multiple document interface is that it allows you to intuitively move and copy objects from one drawing to another. The first part of this chapter discusses how to copy and move objects, including symbols, and scripts, between open document windows.

The remainder of the chapter provides a complete reference to the commands available from the Windows menu.
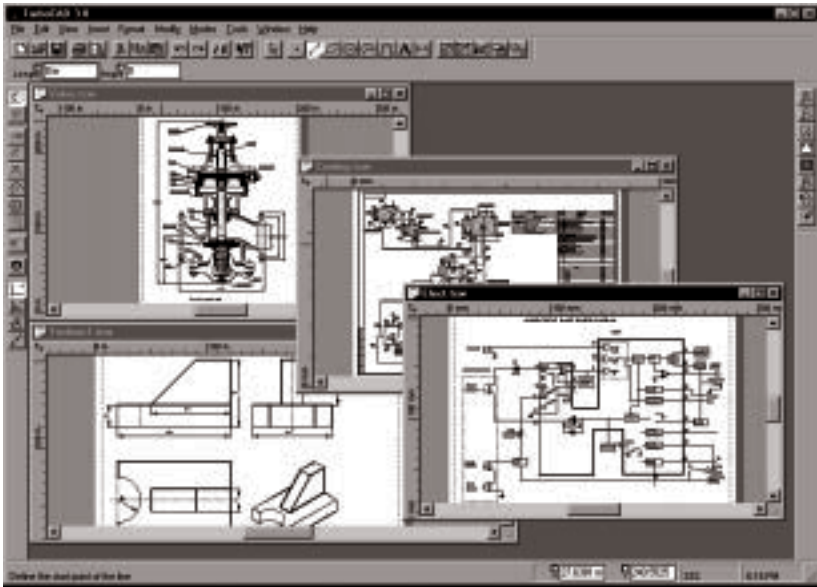
KeyCAD application window containing document (drawing) windows

Moving and Copying Objects between Drawings

There are two basic methods for copying and moving objects between drawings: copy and paste (or cut and paste), and OLE drag and drop.

NOTE: *In describing how to use these techniques, we'll refer to the drawing from which you'll be copying or moving objects as the* source document*; we'll refer to the drawing into which you'll be moving or copying the objects as the* target document*.*

TIP: *Before dragging and dropping objects, it's a good idea to choose the Tile command from the Window menu. This will arrange the windows on the screen so that they are both completely visible, without overlap.*

Copying, Cutting, and Pasting Objects

You can use the Windows clipboard to transfer objects between drawings, regardless of whether both of the drawings are open at the same time. Cut and paste allows you to remove an object from the source document at the same time that you paste the object into the target document.

1. Select the object or objects in the source document that you want to copy. You can use any of the select tools (available from the Select toolbar), or any of the Select commands from the Edit menu.

2. Choose Edit|Copy (<Ctrl>+<C>), or Edit|Cut (<Ctrl>+<X>) from the menu. This places a copy of the objects on the Windows clipboard. (Edit|Cut will remove the selected objects from the source document.)

3. Activate the target document by clicking in its window, or by choosing it from the Window menu.

4. Choose Edit|Paste from the menu, or press <Ctrl>+<V>. The object will be pasted from the Windows clipboard into the target document.

The objects that you pasted from the clipboard will appear at the center of the target document window.

Drag and Drop

OLE drag and drop provides a simple and intuitive way to copy objects between drawings. You can drag objects from one drawing window into another, and you can drag objects stored in symbol libraries from their library palette into another drawing.

Dragging and Dropping between Drawings

To drag a drawing object from one drawing into another:

1. Select the object or objects that you want to drag into another drawing.

2. Place the cursor over the reference point of the selection. This is a movable handle located by default at the center of the selection. You will know that the cursor is in the right place when it turns into a four-directional pointer.

3. Press and hold down the mouse button while dragging the selected objects into the window of the target document. An outline of the selection will follow the mouse cursor as you drag.

4.   Release the mouse button to drop the selected objects. A copy of the selected objects will appear at the point where you released the mouse button.

WARNING: *If you drag an object by any point other than the reference point, you will only be able to drag the object within the window of the source document. Make sure that the mouse cursor is placed over the reference point of the selection before attempting to drag the selection into another window.*

Dragging and Dropping Library Objects between Drawings

You can also drag and drop objects from a drawing's libraries into another document. Moving a symbol to another drawing accomplishes two things: It inserts the symbol into the target drawing, and it places the symbol itself into the corresponding library of the target document.

1.   Open the symbol palette using View|Symbol.

2.   Place the mouse cursor over the symbol in the palette that you want to copy, then hold down the mouse button while you drag the symbol into the target document window.

3.   Release the mouse button. The symbol will appear in the target document at the place where you released the mouse button. The symbol will also appear in the target document's symbol palette.

NOTE: *After you have moved the object, the target document becomes the active window. Notice that the symbol palette will change, displaying the contents of the target document's symbol library.*

Command Reference

The following table summarizes the commands accessible from the Windows menu:

| Command	Function
Tile	Arrange windows so that they are equal-size, side-by-side, with no overlap.
Cascade	Arrange windows so that they overlap with title bars visible.
Arrange Icons	Line up minimized windows along bottom at bottom.
Close All	Close all open windows.
Recent File	Lower part of menu lists open drawings; choosing a drawing makes it the active window.

Tile

Menu: Window|Tile

Use the Tile command to arrange all the open drawing windows into a mosaic. The windows are resized so they all are the same size and fit entirely within the application window.

TIP: *This is a convenient arrangement when you want to compare the contents of two documents or drag and drop objects between documents.*
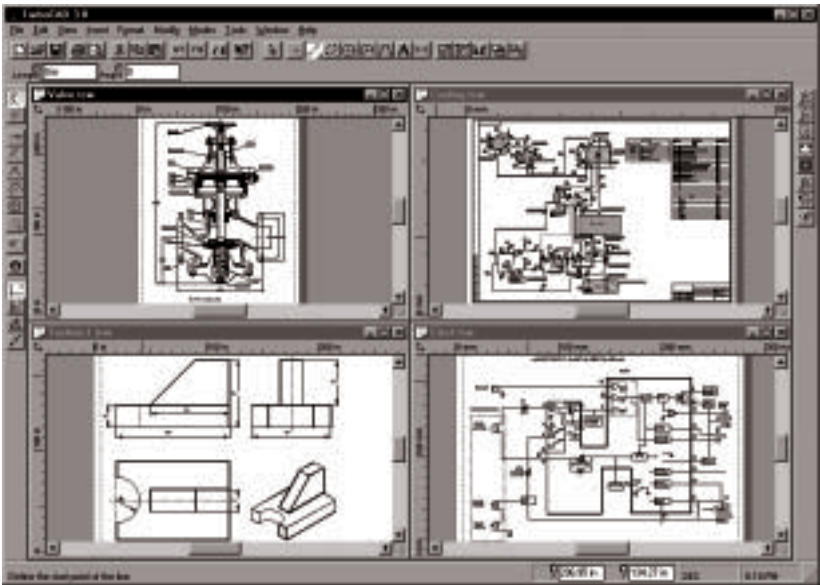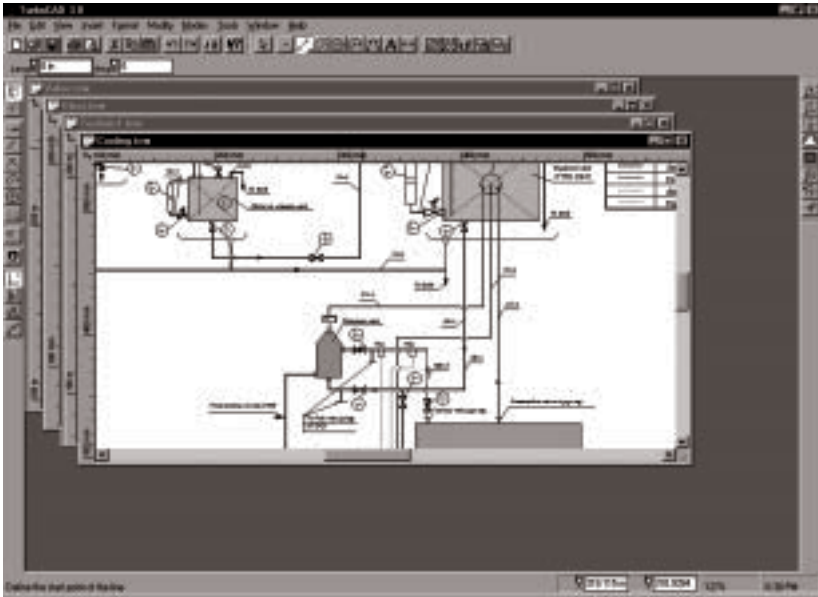

Tiled drawing windows

Cascade

Menu: Window|Cascade

Use the Cascade command to resize and arrange all the drawing windows so they overlap with their title bars showing. Because you can see a portion of each window, you can easily activate any window by clicking it. Only the topmost window is completely visible.

Cascaded drawing windows

Arrange Icons

Menu: Window|Arrange Icons

When you minimize a drawing window, Windows shrinks it to an icon. Use Arrange Icons to line up these iconized windows along the bottom of the application window.

NOTE: *The general shape and appearance of the icon will vary, depending on whether you are using Windows 95 or Windows 3.1.*

Close All

Menu: Window|Close All

Use Close All to close every open drawing window.

NOTE: *KeyCAD will warn you if an open drawing has unsaved changes. Click the Yes button in the warning dialog to save the changes; click the No button to discard the changes. Click Cancel return to KeyCAD without closing the drawing windows.*

Changing the Active Window

KeyCAD lists all of the open drawings in the lower half of the Windows menu. The currently active drawing will have a check mark. When you choose a drawing from the menu, *KeyCAD* makes it the active window. This is the same as clicking in window.

Appendix A

Tips for KeyCAD 2.0 Users

Although *KeyCAD* 3.0 has been redesigned from the ground up, it has retained the features that made *KeyCAD* 2.0 the easiest to use and most intuitive of CAD programs. You'll find enhancements throughout *KeyCAD*, in the form of both new features and improved operation. This appendix points out some of the more important enhancements.

Improved Interface

KeyCAD 3.0's enhanced interface is more customizable, helps you maximize your screen space, and gives you quick access to *KeyCAD* 3.0's larger feature set.

Dockable Toolbars

You can now drag *KeyCAD's* toolbars and place them on your screen vertically, horizontally, or as floating palettes.

Flyout Toolbars

Flyout toolbars make it easier to keep the tool you need onscreen while preserving valuable drawing space. Just click and hold on the category of tool that you want, then choose your drawing tool from the flyout toolbar that appears. Tools that contain flyouts are marked with a yellow triangle.

Local Menus

The local menu, which you access by right-clicking in the drawing area, lets you switch snap modes and other options at any time. The many options that you'll find in the local menu vary depending on the tool you're using (and sometimes even on the location of the mouse), so be sure to check the contents of the local menu as you're familiarizing yourself with *KeyCAD's* new tools.

Property Sheets

KeyCAD's new dialogs contain a wealth of new options and settings that you now access through tabbed *property sheets*. Just click on the tab to bring the property sheet to the front. This is where experienced users can find many of *KeyCAD* 3.0's advanced features.

Drag and Drop Symbols

Symbols in *KeyCAD* 3.0 have been completely redesigned. Instead of using .SLW files, which combine symbols into a single file, you can now treat any file that *KeyCAD* can read as a symbol, including .DXF, .KCW, and even 3D .MDL files from *KeyCAD* 3D. Any folder that contains drawings of the same type can be used as a symbol library.

The new Symbol Palette lets you view thumbnail previews of any symbols, the simply drag and drop them into your drawing. You can also drag and drop objects from your drawing into the Symbol Palette to turn them into symbols.

You can drag and drop symbols into and out of the Symbol Palette

Templates

You can store symbol libraries, drawing setup, scripts, and more in
KeyCAD's new templates. You can use the templates supplied with
KeyCAD to use industry standard drawing setups, or create your own
custom templates: one for each type of drawing that you do.

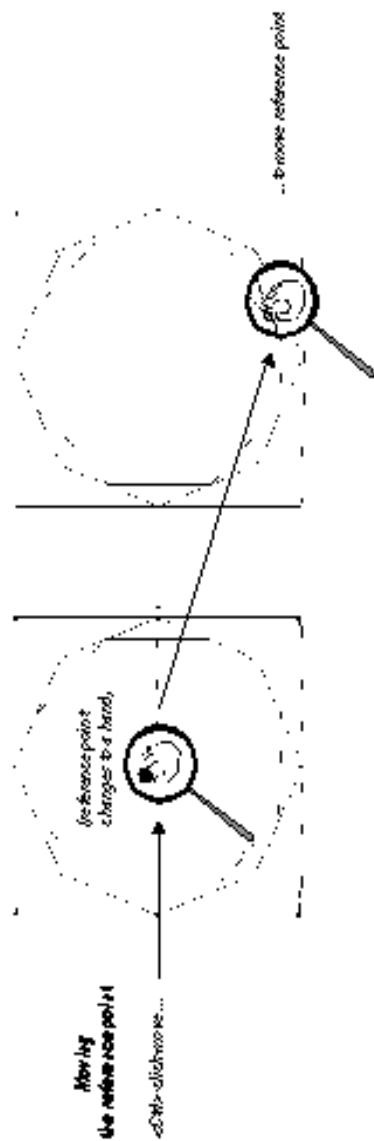SEE ALSO: *Create from Template Dialog*

Improved Programmability

For the programmer, *KeyCAD* 3.0 offers greatly enhanced
programmability. You can write BASIC scripts that call the many
functions exported by the KCAPI or KCAPI32 DLL's, then run them
using the Enable Basic interpreter built into *KeyCAD*. Advanced
programmers can create their own custom DLL's that call the KCAPI
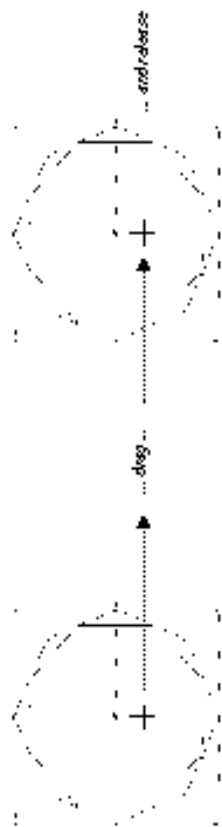functions.

Appendix B

Quick Reference Pages

The Quick Reference Pages that follow summarize the Select Edit and Node Edit techniques. For a complete description of these techniques, see *Chapter 4*.
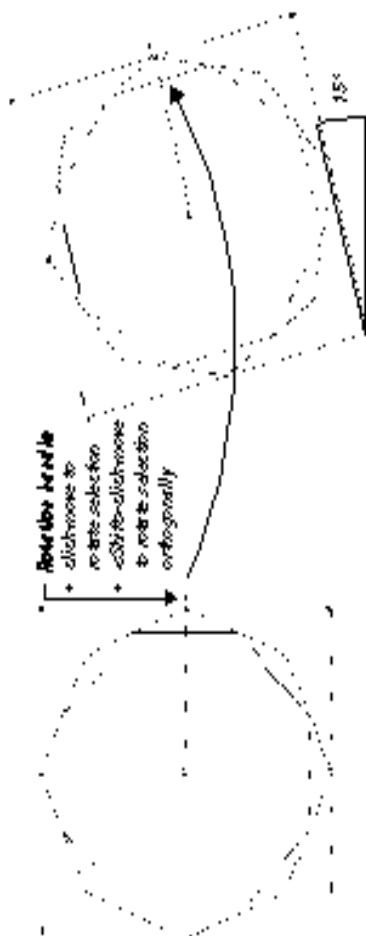
Select Edit

Moving the reference point

<Ctrl>-click-on …

reference point changes to a hand,

… to move reference point

Moving an object

- click on reference point to move selection, using any snap mode or Coordinate Fields
- <Shift>-click-drag to move selection orthogonally
- click-and-drag for OLE Drag & Drop, to drag selection into another drawing or application
- click anywhere in selection …

… drag …

… and release

Select Edit (continued)

Rotation handle
- click/move to rotate selection
- Shift-click/move to rotate selection orthogonally

Resizing with the border
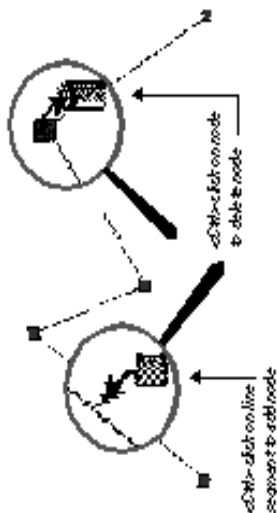- move top or bottom handle to scale height
- move side to scale width
- move corner to scale height and width
- Shift-click move to scale orthogonally
- Shift-drag any to scale proportionally

15°

Other options
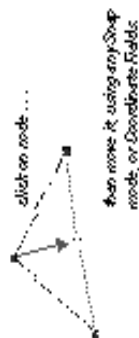- hold down <Ctrl> key after pressing mouse button to copy selection while moving or scaling
- specify Xscale, Yscale, and Rotation in Edit Bar
- with Rubber Stamp selected on the local menu, click to make multiple copies of the selection
- click on a reference point, specify a destination in the Coordinate Fields of the Status Bar, and press <Enter> to move selection

Node Edit Linear Entities

Cancel
Undo Move
Redo   Ctrl+Z / Ctrl+Y
Local Snap
Select Edit
Delete Node
Close
Close Double End
Snap gaps
Drop outline

Closing Double Lines
right-click on endpoint and
choose Close Double End

Cancel
Undo Rotate   Ctrl+Z
Redo Break   Ctrl+Y
Local Snap
Select Edit
Delete Node
Break
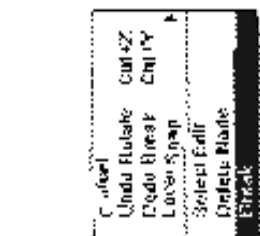
Breaking Polygons
right-click on node
and choose Break

Cancel
Undo Multiline   Ctrl+Z
Redo   Ctrl+Y
Local Snap
Select Edit
Delete Node
Close

Closing Multilines
right-click on endpoint
and choose Close

Adding overlapping nodes

double-click on node
to delete node

double-click on line
segment to add node

Moving nodes

click on node ...

then move it using any Snap
mode, or Coordinate Fields

Cancel
Undo Line   Ctrl+Z
Redo   Ctrl+Y
Local Snap
Select Edit
Add Node
Segment Divide...

Dividing Line Segments
right-click on line segment
and choose Segment Divide

Node Edit: Circles and Arcs

move to change end angle

move to change start angle

move to change radius

Closing Arcs

right-click here and choose
Close the Arc

Cancel
Undo Edit   Ctrl+Z
Redo   Ctrl+Y
Local Snap
Split Edit
Close the Arc

Switching to Node Edit mode

Cancel
Undo Rotated Box   Ctrl+Z
Drag   Ctrl+Y
Local Snap
Node Edit
Make a Copy
Rubber Stamp
Properties
✓ Open Window angle

with entity selected
right-click and
choose Node edit
from the Local menu
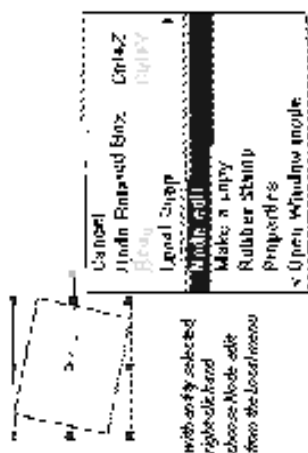
bounding box disappears
only in Node-edit mode

Node Edit: Ellipses and Elliptical Arcs

move this node to
change major axis
<Shift>-drag to
constrain
proportions

move this node to
change minor axis
<Shift>-drag to
constrain
proportions

KeyCAD for Windows