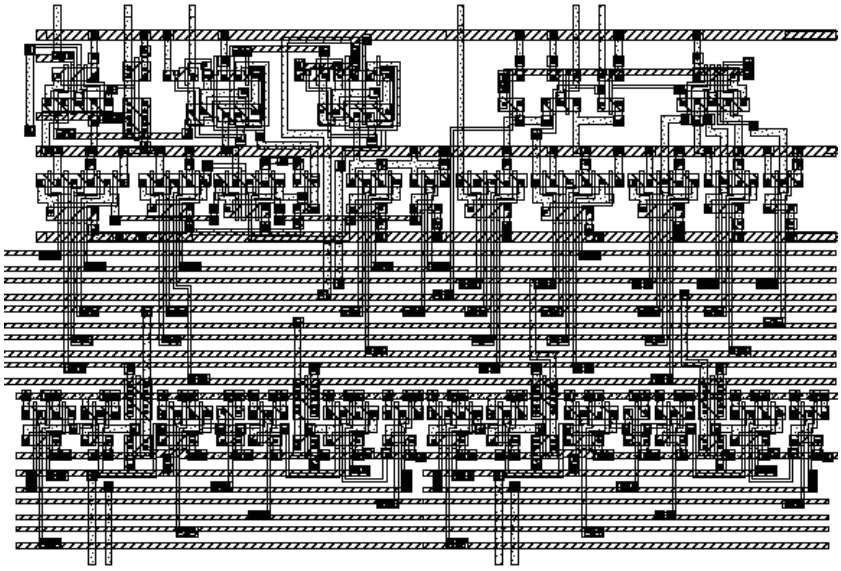
ICED™

Layout Editor for Windows

Basic Tutorial



Version 4.xx

IC Editors, Inc.

The software described in this manual is subject to change without notice. Although all information is given in good faith, neither IC Editors, Inc. nor its agents accept any liability for any loss or damage arising from use of the software or from use of any of the information provided herein.

Acknowledgments:

The majority of this manual was written or revised by Ference Professional Services in Sonora, CA. We are also responsible for formatting the text and creating the screen captures that illustrate the examples.

Michael Gentry of MGC, Inc. created the layout that is used on the cover and as a frontispiece. It is a section of a CMOS simulation of a 74181 4-bit ALU.

Table of Contents

Introduction

This **Layout Editor Basics Tutorial** covers the basic features of the ICED™ layout editor, a full-featured editor for integrated circuit mask sets. This material is intended primarily for new users. The emphasis is on the basics of creation and modification of geometry.

This tutorial is only one of the tutorials from the full ICED™ Layout Editor Classroom Tutorials Manual. You can download the entire manual from the IC Editors website, **www.iceditors.com**. Other tutorials covered in the full manual are:

Customizing ICED™ for Specific Projects will teach you the correct way to set up projects before opening the editor. Lessons will teach you how to create, protect, and share cell libraries. Other lessons cover the creation of technology-specific startup command files that control layer definitions and other environment settings stored in new cell files. We strongly recommend that all ICED™ users become familiar with this information.

The next tutorial, **More on Entering Commands**, covers features that allow you flexibility in how you execute commands. Menus and typed command syntax are covered here in more detail than in the basic tutorial. In addition, keyboard shortcuts and command files are introduced.

All commands that perform operations on existing geometry require that the appropriate component(s) are selected. Methods of selecting entire components, selecting only certain edges or vertices, and preventing certain components or layers from changes are covered in the **Selecting Components** tutorial. Once you have mastered these exercises, you can save large amounts of time in common tasks, since selecting the correct components is often the most time-consuming part of a modification operation.

The **More on Creating and Modifying Components** tutorial begins with lessons that teach you all about layer definition. It continues with exercises that create every type of ICED™ component, including text components, wires, cells, and arrays. All relevant modification methods are also covered including cutting, mirroring, rotating, aligning, and swapping components.

If you use components with sides not at 90º, you should perform the exercises in **Creating Components at Arbitrary Angles**. The creation of circular components, and the modification of shapes with skewed sides is included.

Methods of investigating and modifying nested cells are covered in **Exploring Cell Hierarchy**.

Manipulating the Display covers a variety of methods to change the way data is displayed in the view window. This includes more details on layer appearance such as color and pattern definition. Turning off the display of individual components or entire layers is also covered. Other methods that simplify the display of dense designs are included along with the use of visual aids such as display grids and wiring guides.

Methods of getting useful illustrations of your layout from your printer or plotting device are covered in **Plotting Layout Data**. This includes exercises on the more recent features of the editor that support bitmap export and plots with more than eight colors.

You will create ICED™ cells from sample Stream files and create Stream files from cell data in the lessons in **Importing and Exporting Data with GDSII-Stream Files**. Even if you do not routinely use this industry standard exchange format, you may find the lesson on scaling the size of a design during import, or the lesson on archive backups, relevant to your needs.

Command files are text files containing ICED™ commands. **Creating Useful Command Files** tutorial, you will create commands files that create and modify various types of geometry by manipulating coordinates and components. You will even create a command file that loops through all subcells of a design, snapping all coordinates to the resolution grid. Once you discover how easy it is to write and execute command files with these lessons, you will probably use them more frequently in your work, saving countless hours of tedium.

If you use the DRC program (the Design Rules Checker, available separately from IC Editors), you will find the lessons in **Using the Internal DRC** very valuable. You will perform Boolean operations on selected shapes and entire layers with ease. You will also execute a simple design rules set on a design and step through the errors found without ever leaving the layout editor.

The final tutorial includes a few short exercises that show you how to recover from various sorts of disasters. **Recovering from Mistakes or Crashes** includes information on cell backups and the journal file used to restore cells to a state they were in before a system crash or editing mistake.

Enjoy.

Layout Editor Basics Tutorial

This brief tutorial is intended to acquaint you with the way the ICED™ layout editor operates. It uses the most basic options of the most basic commands to demonstrate the general methods used to create and modify components. The tutorial focuses on using the menus supplied with ICED™ rather than on typing commands at the prompt or using command files.

Starting ICED™

All ICED™ products are designed to be launched from a DOS window. The best way to open a DOS window for this purpose is to use the ICED™ icon. This icon was created on your desktop during installation and displays the representation of a silicon wafer. Using the ICED™ icon sets the system's search path. This ensures that operating system looks in the correct directory for the program executable files. It also sets the current directory to the ICED™ directory, Q:\ICWIN[1]. Double click on the ICED icon now.

Figure 1: ICED desktop icon

ICED™ creates and stores data in cells. Each cell is stored as a separate file. We will use the TUTOR subdirectory to store the cell files created during this tutorial. You generally change to the directory where cell files are stored before launching the layout editor. To change the current directory, type the following command at the DOS prompt in the console window:

CD TUTOR <Enter>

If you have already performed any ICED™-related tutorials, there may be files in this directory left over from a previous tutorial. If this is not the case, or if you want to keep these files, you can omit the next step. However, it is best to start with an empty directory. Delete all files in the TUTOR directory by typing:

DEL *.* <Enter>

[1] Throughout this manual, Q: and \ICWIN are used to represent the drive and directory where you have installed the ICED™ software. If you have installed the software on your C drive in the directory \ICED, you should replace Q: with C: and \ICWIN with \ICED.

ICED™ is usually launched by executing a project batch file that sets several environment variables and ICED™ command line options. The installation provides you with a sample batch file for this tutorial, Q:\ICWIN[2]\ICWIN.BAT. If you use this batch file, all you have to supply is the name of a cell to begin editing.

We will use this batch file to create a new cell with the name "MYCELL". At the DOS prompt, type:

ICWIN MYCELL <Enter>

When ICED™ starts up, it checks to see if the cell you have specified already exists or if it must be created. For new cells, ICED™ executes the startup command file specified in the command line in the batch file. (A listing of this startup command file is provided later on page 38.) This startup command file contains commands that define layer names and properties along with other parameters. After the startup command file is complete, the editor window will be displayed.

The ICED™ Window

After executing the startup command file, ICED™ displays a view window, a menu on the right side of the window, and the command and history lines on the bottom of the window. See Figure 2. An echo of the last command executed from the startup command file will be present on the history line.

The menu cursor indicates the current menu item with a box. In this case, the selected item is **Again**.

Type any letter on the keyboard repeatedly and you will see that you are adding characters at the command line cursor on the command line. The area on the same line to the left of the '>' symbol is called the command prompt. It displays two pieces of information, the current default layer and the number of selected components. Clear the typing on the command line now by pressing both mouse buttons at the same time.

[2] Remember that Q: and \ICWIN are used throughout the manual to represent the drive and directory where you have installed the ICED™ software.

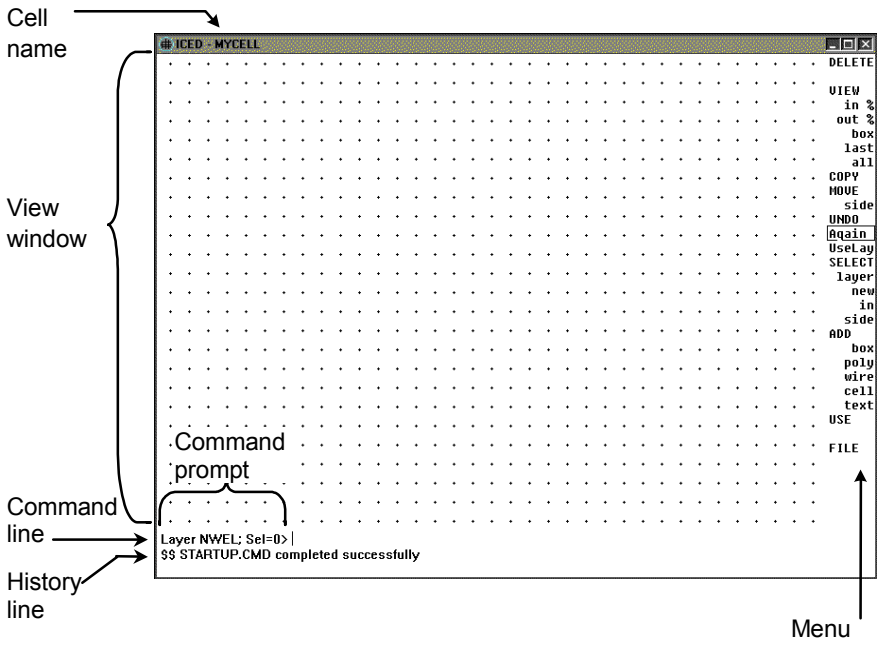


Figure 2: Elements of the ICED™ window.

Closing the Layout Editor

If you want to interrupt this tutorial you can do so by following the instructions in Closing the Editor Normally on page 20. If you are using the free demo version, follow the instructions in Journal Files and Recovery from System Failure on page 19.

Entering Commands

Commands may be executed with any of the following methods:

- Selecting them from the menu using the mouse,
- Typing them on the command line using the keyboard, or
- Executing command files.

In this tutorial, most commands will be selected from the menus using the mouse.

Mouse Functions - An Overview

ICED™ works with either a two-button or a three-button mouse. The functions associated with each button change depending on the operation currently being executed.

In general, the mouse can only be used to select menu items until you finish selecting a command. Once you are executing a command that requires coordinate information, the mouse can only be used to digitize coordinates. Once the command is complete, the mouse can again be used to navigate through the menus.

The more common uses of the mouse are summarized in the simplified tables in Figure 3.

While a menu is displayed	
Left button	Selects menu entries
Right button	Cycles through top-level menus Indicates that you are finished selecting items from a menu list
Both buttons simultaneously	Cancels current menu selection and returns to first top-level menu
Center button	Ignored

While digitizing coordinates	
Left button	Digitizes coordinates
Right button	Indicates that you have finished digitizing coordinates of a wire or polygon
Both buttons simultaneously	Cancels the current command
Center button or <Esc> key	Invokes the nested view menu

Figure 3: Main uses of the mouse buttons

Using the Menus

ICED™ provides you with three top-level menus. Press the right mouse button once to see the second top-level menu. Press it again to see the third menu, and a third time to return to the first top-level menu. Now press the space bar on the keyboard. It also rotates the top-level menus. You can return to the first menu from anyplace in the menu system by pressing the left and right mouse buttons simultaneously. Press them now.

Notice that whenever you return to the first top-level menu, the menu cursor is positioned over the **Again** entry. The menu system generally initializes the cursor position to the same entry every time a given menu is displayed.

Some items on the menu open sub-menus when you select them. For example the ADD item in the first top-level menu has many options. Move the menu cursor until the word "ADD" is selected on the first top-level menu, then click the left mouse button. You can see a list of component types to choose from. One of these component types is "BOX".

We can describe this menu selection with the following shorthand:

1:**ADD → BOX**

The number '1' indicates the number of the top-level menu. In this case the item we want is on the first top-level menu. "ADD" is the item you choose by clicking the left mouse button while the item is selected by the menu cursor. After making this selection, you left-click the item "BOX" on the sub-menu.

Press both mouse buttons now to return to the first top-level menu. Note that "box" is also available right on the first top-level menu under the "ADD" item heading. Both "BOX" menu items access the same ICED™ command, "ADD BOX". The item is repeated on the first top-level menu under the "ADD" heading for convenience since it is used frequently.

We can describe this menu selection with the following shorthand:

1:(ADD)**box**

In this case, "ADD" is only a heading under which you locate the item to select. The item you need to select is in bold type, while the heading will be in normal type and enclosed by parentheses.

We will use this syntax for the rest of the tutorials.

Adding a Box

Using the left mouse button, select the ADD BOX command by clicking on:

1:(ADD)**box**

Now move the cursor around the screen with the mouse. As you move the mouse, the current coordinates of the cursor are displayed below the history line at the bottom of the window. Digitize the first corner of the box by pressing and releasing the left mouse button. Do not hold the mouse button down while you drag the mouse. Now move the cursor to another location on the screen and press the left mouse button again. The box has now been added to the drawing on the default layer. (We'll see how to change the default layer on the next page.)

Changing the View Window

You can use a variety of commands to change the view window. In addition, you can use the arrow keys to pan the view window. Holding down the <Ctrl> key while pressing an arrow key will cause the view window to shift in that direction. Hold down the <Ctrl> key and press the < ↑ > key, then the < ↓ > key. (If commands appear on the command line instead, press the <Esc> key to clear the command area and try the arrow keys again without pressing the <Ctrl> key. If you are using the numeric keypad arrow keys, make sure your keyboard's NumLock light is off.)

The VIEW ALL command changes the view window so that everything in the drawing is displayed. Click:

1:(VIEW)**all**

You can also change the size of the view window by using the zoom commands VIEW IN and VIEW OUT. To zoom out by a factor of 2, click the following options:

1:(VIEW)**out % → 2.0**

Setting the Default Layer

The default layer is the layer to which components are added, unless you override the layer specifically. Use the following menu items now to change the current default layer.

1:**UseLay → M1**

Until you change the default layer to another layer, any component added with the ADD menu options will be added to layer M1.

The UseLay operation continues at this point with an ADD menu allowing you to add a component on the new default layer. (In cases where you do not want to add a component immediately, you can simply return to the main menu by clicking the **MAIN** option, but don't do this now.)

Adding a Polygon

After you use UseLay to choose a default layer, a short ADD menu is automatically displayed. Select **POLY** from the menu. The menu will disappear. The program is now waiting for you to digitize the points of a polygon.

Move the cursor to the upper right area of the window and press the left mouse button. Notice that the first point of the polygon still has an X marking it as the starting point of the polygon. Continue to select polygon vertices.

Notice that if you draw a line that doubles back on itself, the doubled over section will disappear. Thus, if you draw a side that is too long you can correct it by doubling back and redigitizing the vertex in the correct position. If the side is too short you can extend it. This does *not* add extra vertices to a polygon.

Finish the polygon by adding vertices until it is closed. The best way to tell ICED™ that you have completed the polygon is to redigitize the starting vertex. Be sure to align the cursor carefully over the X marking the starting vertex. You can also tell ICED™ to close a polygon by pressing the right mouse button.

Repeating, Canceling and UNDOing Commands

Select the following item from the menu:

1:**Again**

This will repeat the last command. The "ADD POLY" command is shown on the command line and the program is waiting for you to digitize another polygon. Instead of adding another polygon, press both the left and right mouse buttons simultaneously to cancel the command.

Another way to repeat commands is to use the arrow keys to select any command that has been executed recently in the current session. Press the < ↑ > key repeatedly until the "ADD BOX" command is shown on the command line. Now press <Enter>.

You are now creating another box. Digitize two points with the left mouse button to define the box. Note that it is created on the new current default layer rather than the layer used when you defined the first box.

Now remove the box you just added to the drawing by clicking:

1:**UNDO**

Executing **UNDO** will usually undo the last non-view command. (The 1:(VIEW)**last** menu item can be used to undo view commands. The two classes of commands are handled separately so that you can execute a command, modify the view window to examine the results, and then decide to undo the command.) UNDOing a command twice redoes the command. Click 1:**UNDO** again now. The box is recreated in the drawing.

Adding a Wire

To add a wire to layer POLY, click:

1:**UseLay → POLY → WIRE**

The menu will disappear and a red octagon with the cursor in the center will appear on the window. This is the wire cursor. Move the cursor where you want the first vertex of the wire to be placed and press the left mouse button.

Now move the cursor around in a circle without pressing any buttons and note that the wire snaps to angles that are multiples of 45°. There are snap grid and snap angle parameters that control how you can digitize points with the cursor. (These parameters are set in the startup command file described on page 38.)

Move the cursor to position the next vertex of the wire and press the left mouse button again. Now move the cursor about 1 inch past the edge of the view window. The view window will pan by one half the view window dimension.

(This feature is called autopan. Autopan is active whenever it is enabled and you are digitizing points for any command other than a VIEW command.)

Continue defining wire vertices. You can correct digitizing errors by doubling back or extending segments. After pressing the left mouse button to place the final vertex, press the **right** mouse button to tell ICED™ you have finished adding the wire.

Entering Commands on the Command Line

While the menu is shown, press the < ↑ > key once. If the "**Use Layer POLY; add wire**" commands do not appear on the command line, keep scrolling the command history with the arrow keys until they do.

The <Home>, <End>, < ← >,and < → > keys move the text insertion cursor on the command line. Press <End> now and type at the keyboard, " **width = 4**". The entire command should now look like the following:

Use Layer POLY; add wire width = 4

Press <Enter> now. You are now adding a wire with a different width. Press both mouse buttons to cancel the command.

You can type a command at the prompt any time a menu is displayed. Any input from the keyboard is automatically typed on the command line. You do not have to move the cursor to any particular location.

The SELECT IN Command

Many ICED™ commands operate on selected components and ICED™ offers a variety of ways to select them. The **SELECT IN** command selects any components whose outlines cross or are completely inside of a specified box. Select this command now by clicking:

1:(SELECT)**in**

Draw a box with the cursor that intersects one of the components that you have already created. To do this, move the cursor to a position just outside the shape and press the left mouse button. Then move the cursor so that it is inside the shape and press the left mouse button again.

When an item is selected, select marks (small white squares) are drawn on the boundaries of the component. Notice that the number of selected components is shown in the command line prompt near the lower left corner of the window after the text "Sel=".

More on Menus

When one or more components are selected, the first top-level menu is changed. Notice that a variety of UNSELECT commands, indicated by the heading UNSEL, have replaced the lower case items under the ADD heading. All of the ADD options that are now missing from the top-level menu are still available by clicking 1:**ADD**.

Copying a Component

Now that you have selected a component (it is okay if you have selected more than one), you can copy it (or them) by using the COPY command. Optionally, you can mirror components as you copy them. To perform a copy with mirroring, click:

1:**COPY → MIR Y**

The COPY MIRROR Y command will mirror the component(s) around a horizontal line drawn at a particular Y coordinate. Use the mouse to position the horizontal line and press the left mouse button to perform the copy operation.

Unselecting All Components

Now select the menu item:

1:(UNSEL)**all**

All components are now unselected.

Embedded Select Commands

The sequence of actions consisting of selecting a group of components, operating on them, and then unselecting them is very common in ICED™. Any number of components can be selected at one time. You can build a large set of selected components by combining a series of SELECT and UNSELECT commands. Furthermore, you can perform several commands on the same set of components before unselecting them. However, in simple cases, the select-operate-unselect sequence is unnecessarily cumbersome.

If no components are selected and you issue a command that operates on selected components, ICED™ will give you an opportunity to select something before executing the command. This is called an embedded select command. Embedded select commands combine the SELECT and UNSELECT commands with the command itself.

To see how this works, make sure that no components are selected by noting the "SEL=0;" remark in the command line prompt. (If a number other than 0 is reported, click 1:(UNSEL)**all**.) Click:

1:**MOVE → X&Y**

A cursor with a box will appear in the view window. Use the mouse to position the cursor box so that it crosses one side of the wire you added earlier. Press the left mouse button. The wire should now be selected and the cursor will change back to the normal cursor.

You must tell ICED™ how far to move the component by digitizing two reference points. ICED™ will move the component by the distance between them.

You already digitized the first point when you selected the component. Note that this point is marked with a white X. (If the first point is in an inconvenient location you can cancel it by pressing the right mouse button.) Move the mouse and press the left mouse button to digitize the second reference point. ICED™ will move and then unselect the wire.

Because this tutorial deals with simple geometries, you will usually use embedded select commands. It is important to remember that you can always use the select-operate-unselect sequence in more complicated situations.

Note: The exact nature of the embedded select command depends on the command you are using. In this case you used an embedded SELECT NEAR command and the box cursor is called a “near-box”.

Selecting and Stretching Parts of a Component

In addition to manipulating entire components, ICED™ can manipulate portions of a component. This is done by selecting only the sides or segments of the component you want to transform. Click:

1:(SELECT)**side**

Draw a box that intersects only one segment of the wire you have already added. You have now partially selected the wire. Move the selected segment to a new position by clicking:

1:**MOVE → X&Y**

The selected segment and the segments attached to it will be stretched or shrunk to place the segment in its new position. Move the cursor and select a point on the selected wire segment and press the left mouse button. Now move the cursor to the new position for the segment and press the left mouse button again.

To unselect the wire, click:

1:(UNSEL)**all**

You have just finished a select-operate-unselect sequence. You could have done the same thing with an embedded select command. Do this now to a different wire segment by clicking:

1:(MOVE)**side → X&Y**

Journal Files and Recovery from System Failure

Every time ICED™ completes a command, the command is recorded in the file *cell_name*.JOU in the working directory. (In this case, "Q:\ICWIN[3]\TUTOR\-MYCELL.JOU".) When you use the normal termination commands (EXIT, QUIT, or LEAVE) to terminate ICED™, this file is renamed *cell_name*.LOG.

Whenever ICED™ is launched to edit a cell, it looks to see if the file *cell_name*.JOU exists. If it does, ICED™ knows that your last session was interrupted and asks you if you want to recover. If you indicate that you want to recover your work, ICED™ will re-execute the commands in the journal file. This process is very fast, since ICED™ will not regenerate the display for each command.

[3] Remember that Q:\ICWIN represents the drive and path where you have installed ICED™.

Click the button with the 'X' in the far upper right hand corner of the window. A message box is displayed to see if you really want to close the editor without saving the cells files. Click the button labeled "JOURNAL" to proceed without saving the work you have done. This is similar to the situation you face in a power failure or when you turn off your computer without closing the editor.

Now edit cell MYCELL again. In the console window opened by the ICED icon, you can simply press the < ↑ > key to recall the last command on the command line. When "ICWIN MYCELL" is on the command line, just press <Enter>. ICED™ will display a message similar to:

Journal file for Q:\ICWIN\TUTOR\MYCELL exists
Do you want to recover?

Click the "YES" button. Note that the prompt at the bottom of the window tells you to press <Enter> to continue. Do so now. The journal file is executed, recovering all work you did up to the point where your session was interrupted.

(Far more complex recovery scenarios are possible with this type of recovery mechanism. An entire tutorial covers this subject in the Classroom Tutorials Manual.)

Closing the Editor Normally

If you are using the free demo version of ICED™, the journal method described in the preceding lesson is the only method available to you to save your work. This version will not save cell files. If you close the edit session with the 'X' button in the far upper right hand corner of the window (or use the JOURNAL command), you will be able to recover all of your work the next time you open the same cell.

The purchased version will save cell files. To close the editor and save all work done during the session, use the LEAVE command. To test this command, click:

1:**FILE → LEAVE**

Either method will leave the console window created by the ICED icon open. If you need to close this window now, type "EXIT" at the console command prompt or click the Close button (with the X icon) in the top-right corner of the console window.

You have completed about one third of the tutorial. This is a good point to take a break.

To open the editor again, follow the steps on page 7. If you left the DOS console window open, you can simply press the < ↑ > key to recall the "ICWIN MYCELL" command and then press <Enter>.

Using the Ruler

During layout, there will be times when you need to measure the distance between two objects. This is done with the RULER command. Click:

2:**RULER**

(Remember that you must press the right mouse button (or space bar) to change to the second top-level menu. Select the **RULER** item near the middle of the second menu.)

A full window cross hair will appear. Move the intersection of the cross hair to the place you want to use as your starting point and press the left mouse button. Now, move the cross hair around using your mouse. Notice that at the bottom of the window, the current location of the cross hair, the position of the starting point, the X and Y displacements, and the diagonal distance between the two points is displayed. Press the left button again to end the use of RULER.

Nested View Commands

When editing large designs the limited resolution of the screen becomes a problem. Nested view commands provide a method of easily changing the view window from a large-scale view to a view that displays fine details during the execution of commands. You sometimes need to view a large section of the design, select a command from the menu, zoom in on a small section of the design, select a point, return to the original view, zoom in on another section of the design, select another point, and so on.

To see how this works, we will add a new box at a distance from the current geometry, then measure this distance using the RULER command and the nested view menu. First, click:

1:(VIEW)**out % → 5.0**

Add a box at the lower left corner of the view window with the menu option:

1:(ADD)**box**

To perform the measurement, click:

2:**RULER**

The RULER cross hair will appear, however at this scale it might be difficult to accurately measure the distance from the box to the other components.

To zoom in on the lower left corner of the view window, press the **<Esc>** key. (The center button on a three-button mouse performs the same function.) A VIEW menu with a red cursor will appear. Select **BOX**. Draw a small box that surrounds the new box in the lower left corner. Once you have digitized the view box, the window zooms in and the cursor returns to the RULER cursor allowing you to continue the interrupted RULER command. Now you can accurately place the cross hair on one corner of the box. Do so and press the left mouse button.

To return to the view at the start of the command, press the <Esc> key (or center mouse button) again. This time select **HOME**. This returns the view window to what it was when you began the current edit command.

Now press the <**Esc**> key (or center mouse button) again and use **BOX** again to zoom in on the components in the center of the window. Place the cross hair on the corner of one of the components and press the left mouse button. This completes the RULER command.

To return again to the view at the start of the RULER command, select:

1:(VIEW)**last**

Nested view commands are useful in a variety of situations. The <Esc> key (and center mouse button) are active whenever you are selecting points for any command other than a VIEW command.

If you click the left and right mouse buttons simultaneously during a nested view command, you will only cancel the nested view command.

Deleting Selected Components

The **DELETE** command removes fully selected components from the layout. (It does not affect partially selected components.) To clear the layout, click the following menu items:

1:**SELECT → ALL**
1:**DELETE**

Now select **UNDO** from the main menu. The components will reappear. Select **UNDO** a second time to redo the DELETE command.

The TEMPLATE Command

In the exercise on page 14, you added a wire using the default width for the layer POLY. This default width parameter was set by the startup command file. You can use the TEMPLATE command to display a listing of the ICED™ parameters, including the default width for each layer. Execute this command now by clicking:

2:(TEMPLA)**screen**

A listing of ICED™ parameters will replace the view of the layout. If you do not see a group of lines all beginning with the keyword "LAYER", as shown in Figure 4, use the mouse on the scroll bar on the right edge of the window (or the < ↑ > and < ↓ > keys) to scroll through the listing until they appear.

```
LAYER 0   PEN=0
LAYER 1   NAME=NWEL     WIDTH=3.000     DIM_BLUE        PAT=1   PEN=16 …
LAYER 2   NAME=NDIF     WIDTH=3.000     GREEN   PAT=1   PEN=* …
LAYER 3   NAME=PDIF     WIDTH=3.000     YELLOW  PAT=1   PEN=* …
LAYER 4   NAME=PSEL     WIDTH=3.000     YELLOW  PAT=0   PEN=* …
LAYER 5   NAME=POLY     WIDTH=2.000     RED     PAT=1   PEN=* …
LAYER 6   NAME=M1       WIDTH=3.000     CYAN    PAT=2   PEN=* …
```

Figure 4: Portion of TEMPLATE report

Part way down this list, you will see an entry with the parameter "NAME=POLY". This line displays the parameters associated with the POLY layer. Note the default width indicated by the "WIDTH=" keyword.

When you are done viewing the report, press <Enter> (or both mouse buttons simultaneously) to return the view to the layout mode.

Adding a Wire with a Non-Default Width

You can override the default width of a new wire using menu options. First, adjust the view scale so that the red grid appears. To do this, click:

1:(VIEW)**in % → 1.5**

This will zoom in 1.5 times the current scale. Keep clicking 1:**Again** until the red grid appears.

Please add the next two wires carefully. You will need them for the next several exercises. When you are done with this exercise your display should contain wires similar to the two wires shown in Figure 5.

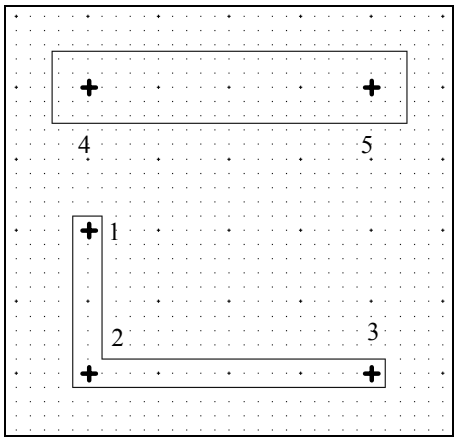


Figure 5: Adding wires.[4]

The ADD command we are about to use will add a wire on the default layer. The default layer name is always reported in the command prompt at the lower left corner of the window. Note what the default layer is right now. If it is not POLY, change it by typing the following command :

USE LAYER POLY <Enter>

Now use the ADD WIRE command to add a wire to that layer using the default width associated with it. Click:

1:(ADD)**wire**

The wire cursor appears, ready for you to digitize the coordinates in Figure 5. Move the cursor to point 1 and press the left mouse button. Then do the same for points 2 and 3. Finally, press the right mouse button to indicate that you are done adding coordinates.

Next, we will override the default layer and width. Click:

1:**UseLay → M1 →** (WIRE)**w% → 5.000**

[4] These graphics are drawn with FILL=OFF. Your display may fill in the outlines with color. Fill is covered later.

This will add a wire on the M1 layer with a width of 5.000. When the wire cursor appears, digitize points 4 and 5 (as shown in Figure 5) then press the right mouse button.

Note that the default layer, as reported on the command line, has changed to M1.

Displaying Component Information with SHOW

The SHOW command can be used to display data about selected components. Start by selecting the two wires you just added to the layout. To do this, click:

1:(SELECT)**in**

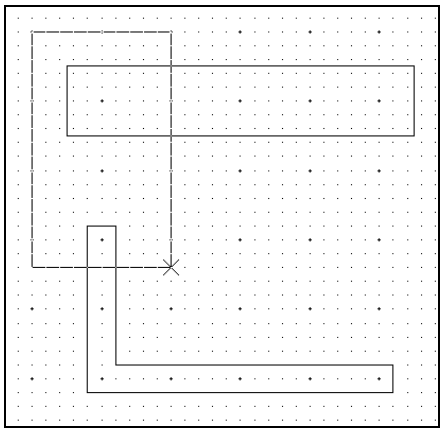


Figure 6: SELECT box used to select 2 wires.

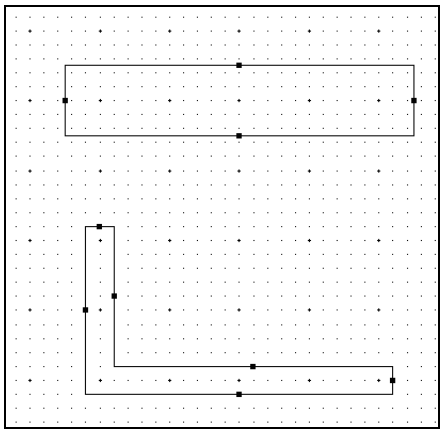


Figure 7: The 2 wires are selected.

Now, digitize a select box intersecting both wires as shown in Figure 6. White select marks will appear on the wires as shown in Figure 7. To display information on the selected components, click:

2:(SHOW)**screen**

The text for the ADD commands of the selected components will appear. (Microwave engineers should note that total wire length is reported.) Note that the LAYER and WIDTH of the two wires are different.

Press <Enter> to return the view window to layout mode.

Unselect the wires by clicking:

1:(UNSEL)**all**

Quickly Reporting the Width and Length of a Wire

The combination of an embedded SELECT command with the SHOW command allows you to display the length and area of a single wire very efficiently. Now that nothing is selected, execute the SHOW command again by clicking:

2:(SHOW)**screen**

The near-box cursor will appear in the view window. Move the cursor so that it overlaps the edge of the **'L' shaped wire**. Now press the left mouse button to select the wire and generate a display similar to Figure 8. **Note the width of the wire** as displayed on your window. (You will need to know this value in the next exercise.) Press <Enter> to return to the layout mode. Note that the wire is unselected at the end of the command.

```
! Selected components in MYCELL:
ADD WIRE LAYER=POLY ID=9 TYPE=2 WIDTH=2.000 AT (69.5, 44.0) &
        (69.5, 14.0) (125.0, 14.0)
! layer:   perimeter     area        wire length
!- POLY  191.000       452.50      90.500
```

Figure 8: Report from the SHOW command including wire area and length.

Changing the Width of a Wire with @ED

In a later exercise, we will be merging the two wires. To merge wires, they must be on the same layer and have the same width. In this exercise, we will change the width of the horizontal wire to be the same as the 'L' shaped wire.

We will do this with the Q:\ICWIN[5]\AUXIL\ED.CMD command file supplied with ICED™. This command file can be used to modify any of the parameters of a given component. A command file contains a list of ICED™ commands. These command files are executed with the *@file_name* command.

Type the following to execute this command file:

@ED <Enter>

The near cursor appears in the view window. Position the mouse on the edge of the **top wire**, then click the left mouse button to select it.

The Windows Notepad editor comes up in a new window displaying the text of the ADD command for the component. (This was generated by a SHOW command in the command file.) Now edit the WIDTH= parameter to change it to the same value as the width of the 'L' shaped wire reported in the last exercise.

To exit the text editor, type <Alt><F> and then <X>. Type <Enter> at the "Save it now?" prompt. This saves the file. The ED.CMD command file will now delete the original component and add a wire component using the ADD command you just edited. Note that the width of the two wires is now the same.

Using @UNED

After you execute a command file, the UNDO command will undo only the last command executed by the command file. To undo the effect of the entire ED.CMD command file you need to use a different method.

Type the following to execute the UNED.CMD command file:

@UNED <Enter>

[5] Remember that Q:\ICWIN represents the drive and path where you have installed ICED™.

You can see that the wire is now back to what it was before ED.CMD was executed. To redo the changes to the wire, execute UNED.CMD again by clicking:

1:**Again**

You can also execute these command files (or any command file stored in a directory known to ICED™) from the menus. You could run UNED.CMD by making the following menu choices to select the command file from lists of command files available in the directories known to the editor:

3:**@%.cmd → NextPATH →** (don't click now) **UNED**

Rather then re-execute the UNED.CMD command file, simply return to the main menu by pressing both mouse buttons.

Changing a Layer with the SWAP Command

In this exercise we will change the layer of the top wire, so that both wires will be on the same layer. We could have done this at the same time we changed the width, however, this method demonstrates a easier way to edit a component (or a selection of components) when all you want to do is change the layer.

The easiest way to change the layer of a component is to use the SWAP command. Click the following menu options:

2:(SWAP)**layers → POLY → M1**

The menu disappears and the near cursor appears in the view window for you to select the horizontal wire at the top of your window. Once you select the wire, its layer is changed from M1 to POLY. You can test this with the SHOW command if you want to.

(The SWAP command is more powerful than a simple "change layer" operation. When components on both indicated layers are selected, they all will have their layers swapped. You can also swap cells with options of the SWAP command.)

Now that both wires have the same width and are on the same layer, we can merge them.

Merging Two Wires

The MERGE WIRES command connects two selected wires to form a single wire. **Both wires must be on the same layer and have the same width.**

In order to merge two wires, you must **partially select** them. Each wire must have one selected end and one unselected end. The selection of the other sides of the wires is not important.

Your display should look something like Figure 5 on page 24. Make sure that no components are selected, then perform the merge with the following menu options:

1:(UNSEL)**all**
2:(MERGE)**wire**

You are now executing an embedded SELECT END IN command. Use the mouse to digitize the same box shown in Figure 6 on page 25. However, since this is a SELECT END IN command rather than a SELECT IN command, only the ends of the wires in the box are selected.

The MERGE command will extend, or contract, the selected ends of both wires and merge them to form a single wire as shown in Figure 9.

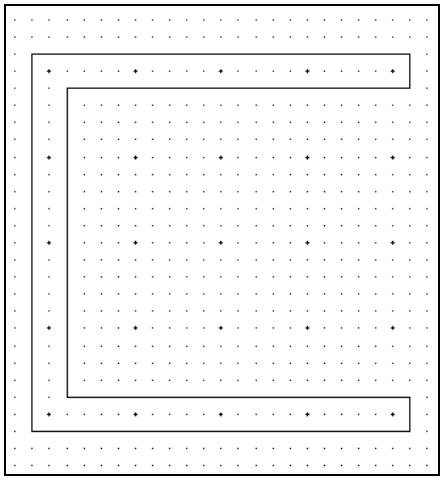


Figure 9: Merged wire.

It is not always possible to select both wire ends at the same time without selecting an extra component. If this is the case, you must select them prior to executing the MERGE command. To see how this is done, select 1:**UNDO**. Then choose 1:(SELECT)**end**. Use the mouse to digitize a box that surrounds the top of the L-shaped wire. This will select the top end of the wire. Now select 1:**Again**. Use the mouse to digitize a box that surrounds the left side of the horizontal wire. Now select 2:(MERGE)**wire** again. Watch what happens.

The FILL Command

ICED™ has two display modes, FILL ON and FILL OFF. You can toggle between them by selecting 3:(FILL)**tog** or you can just type the following abbreviation for the FILL command at the command prompt:

F <Enter>

When you type commands at the prompt, you can always abbreviate keywords as long as the abbreviation is unambiguous. No other command begins with the letter 'F' so typing simply <F><Enter> is sufficient to execute the FILL command.

Click 1:**Again (**if necessary) to toggle the fill mode to off.

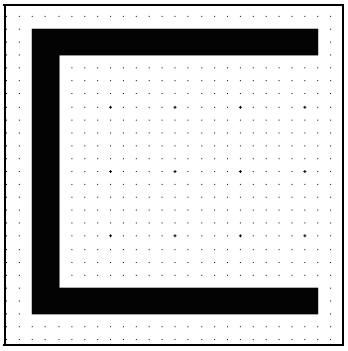


Figure 10: Wire drawn with FILL on.

Merging a Wire and a Box

The MERGE WIRES command can also be used to merge a wire and a box, as long as they are on the same layer and the box has the same width as the wire. The wire must have one end selected and the box must have either one or three sides selected.

To demonstrate this type of merge, we first need to add a box with the appropriate width. Click:

1:**UseLay→ POLY → BOX**

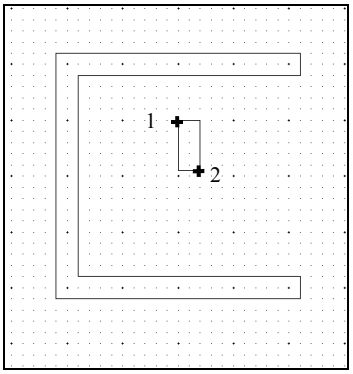


Figure 11: Adding box.

Digitize the coordinates carefully with the cursor. See Figure 11. Use the report on coordinates at the bottom of the window as you are digitizing point 2 to insure that the width of the box is exactly the same as the width you noted in the earlier exercise.

Now we will merge this box with the wire. Click:

1:(SELECT)**side**

Now use the cursor to define a selection box that intersects one end of the box and one end of the wire. See Figure 12. It is important that you partially select only one end of each shape. To perform the merge, click:

2:(MERGE)**wire**

The result of this operation is shown in Figure 13. Note that the end of the top wire segment has been trimmed back to perform the merge.

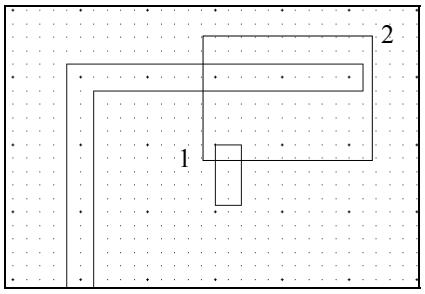


Figure 12: Selecting end of box and wire.

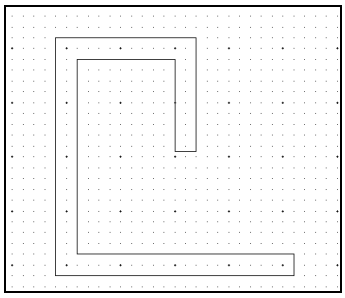


Figure 13: Result of merge.

Cutting a Wire

The CUT command is used to divide wires, polygons, or lines into two parts. To see how the CUT command works, click:

2:(CUT)**hori-Y**

You are now executing an embedded SELECT SIDE IN command. Select only the single vertical segment of the wire as shown in Figure 14.

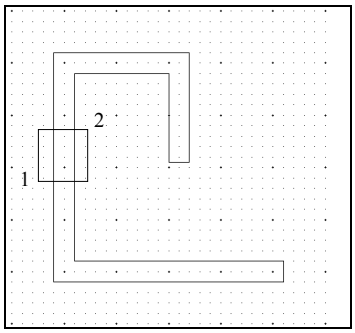


Figure 14: Selecting a segment of a wire.

A horizontal cut-line will appear. (A horizontal line is defined by the equation Y=*constant*. This leads to the menu label, hori-Y.) The wire will be cut where the line intersects the selected segment of the wire. Position the cut line with the cursor so it intersects both vertical segments. Now click the left mouse button.

Even when the cut line intersects more than one wire segment, only the selected segment is cut. If the cut line intersects more than one selected segment, the cut will fail.

Merging Polygons

The MERGE POLYGONS command is used to merge two polygons to form a single polygon. The two polygons **must touch** along at least one side. Either or both polygons may actually be boxes. If the polygons touch each other from the outside, the merged polygon is the sum of the two polygons. If they share a common area as well as a common side, the merged polygon is the difference of the two polygons. Thus, the MERGE command can be used to add or remove pieces of an existing polygon.

Hold the <Ctrl> key and press the < ↑ > key twice to move the view window to a clean section of the drawing. Next, add two boxes as shown in Figure 15 with:

1:(ADD)**box**
1:**Again**

To perform the merge operation, click:

2:(MERGE)**poly**

With the near-box cursor, select the common side as shown in Figure 15. The merged polygon should look similar to the one shown in Figure 16.

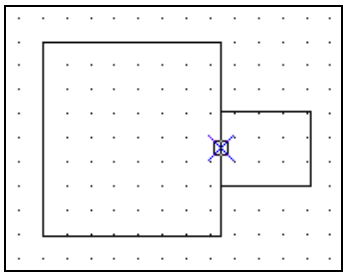


Figure 15: Selecting the common side of 2 boxes.

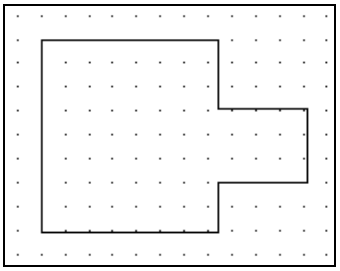


Figure 16: Merged polygon.

Cutting a Polygon

The CUT command can also be used to cut a polygon. To perform this operation, click:

2:(CUT)**vert-X**

Digitize a select box that cuts the top and bottom sides of the polygon as shown in Figure 17. A vertical cut-line will appear and 2 sides of the polygon will be indicated with select marks as shown in Figure 18. The cut-line must intersect exactly two selected sides of a polygon. If it intersects more or fewer sides, the cut will fail. Press the left mouse button to digitize the line position and cut the polygon.

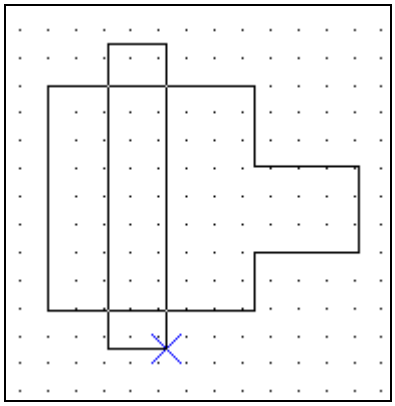


Figure 17: Selecting sides of a polygon.

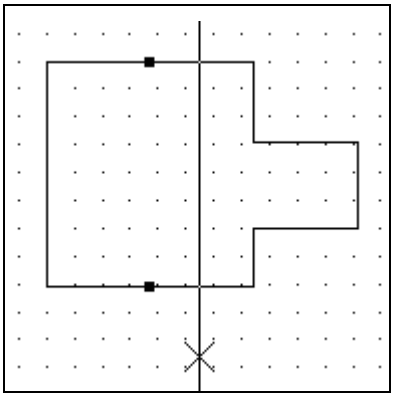


Figure 18: Using cut line to cut the indicated sides of a polygon.

Adding Cells

Hold the <Ctrl> key and press the < ↑ > key twice to reach an unused area of the drawing. Now click:

1:(ADD)**cell**

A cell menu will appear. This menu should list the sample cells in the directory, Q:\ICWIN[6]\SAMPLES. The cell directory name is reported on the history line. (If the SAMPLES directory is not indicated, select the **NextPATH** menu item until it is.)

Select the cell named VIA by moving the cursor to the line **VIA** and pressing the left button on your mouse. After selecting the cell name, you will see the outline of a box displayed on the window. This is an outline of cell VIA. Move the mouse to position the cell then press the left button to place the cell and end the command.

Now let us add another cell. Select the ADD CELL command again:

1:(ADD)**cell**

Note that the first cell list displayed has changed. If you already have cells added to the current drawing, ICED™ will display those cells in the first list. To see cells in the next cell library, select **NextPATH.**

If the Q:\ICWIN\SAMPLES directory is not currently listed, select **NextPATH** from the menu until it is. Now select **NAND**. A cell outline will appear. Press the left mouse button to add the cell to your drawing.

Now let us add another copy of the NAND cell, but we will mirror this copy as we add it. Select the ADD CELL command with the mirror option by clicking:

1:**ADD → (Cell %)mY → NAND**

To add more copies of the same cell, you can use the 1:**Again** menu option. Add a few more copies of this cell. Change the view menu during the command if necessary with the nested view menu by pressing <Esc> or the center button on a three-button mouse.

Ungrouping a Cell

A cell is just a group of components. A cell can be ungrouped so that you can select and modify individual components without modifying other copies of the cell. To see how this works, click:

2:**EDIT → UnGrp**

[6] Remember that Q:\ICWIN represents the drive and path where you have installed ICED™.

You are now executing an embedded SELECT CELL * AT command. Move the cursor inside the white dotted line that surrounds a copy of the NAND cell. (When using the SELECT CELL * AT command, placing the cursor on the edge of a cell will not select it.) Press the left mouse button to ungroup the cell. The dotted white cell outline will disappear. This indicates that the individual components can now be operated on like any other component in the drawing. The other copies of NAND are not affected.

Selecting New Components

The SELECT NEW command generally selects components that were changed by the last non-VIEW command. Select:

1:(SELECT)**new**

Notice that the components of the ungrouped cell are selected.

Unselecting Components by Layer

You can select/unselect components by layer. We will unselect components in our ungrouped cell except for the M1 shapes. Click:

1:(UNSEL)**layer → M1**

Note that the selected layer name turns yellow. You could continue to select more layers at this point. Now click:

Invert

Now the M1 layer name returns to a green color, and all of the other layer names turn yellow indicating that all layers except for M1 will be unselected. To indicate that you have finished selecting layers and continue the command, press the **right** mouse button (or click Return). Select **ALL** from the next menu.

Constraining Moves to one Direction

Move the M1 components in the ungrouped cell down slightly with the option:

1:**MOVE → Y**

This allows the selected components to move only vertically. Now you must digitize two points to define a displacement vector. Place the cursor anywhere on the screen and press the left button once. Now move the cursor down slightly. Note that the exact displacement is reported at the bottom of the ICED™ window in the following format:

(*current location*) -(*first coordinate*) = (*disp_x*, *disp_y*)

Now press the left mouse button to digitize the second coordinate of the move. Note that only the selected components have moved. Note also that the components moved straight up or down. The displacement in the x-direction was ignored.

Using GROUP to Create a Cell

The GROUP command takes fully selected components and creates a new cell from them. The selected components are removed from the drawing, and the new cell is added in their place.

First, we must select all components in our modified NAND circuit. We will use the SELECT IN command for this. Click:

1:(SELECT)**in**

Use the cursor to draw a select box around the modified NAND circuit. This will select the unselected components again. Components that were already selected are not affected.

To remove these components from the layout and add a newly created cell in their place, select:

2:**EDIT →** (GROUP)**at ***

Note the prompt near the bottom of the window and type the cell name as "NANDTEST". Then press <Enter>. You have now created a cell with the name NANDTEST. The origin of this new cell is at the lower left-hand corner of the bounding box indicated by the dotted white line around the cell.

The new cell file (NANDTEST.CEL) will be stored in the working directory, Q:\ICWIN\TUTOR, when you exit the editor (unless you are using the demo version of ICED™).

Aligning Cells

Since our modified NANDTEST cell has had its buss wires shifted, it will need to be aligned with the busses in other cells for the busses to line up. We can align components easily with the MOVE Y (or MOVE X) commands.

First select the entire cell by clicking:

1:(SELECT)**new**

Now begin the MOVE operation with:

1:**MOVE → Y**

Use the cursor and the left mouse button to digitize a reference point on the lower edge of the lowest M1 wire in the NANDTEST cell. Now digitize the second reference point on a lower edge of a similar wire of one of your copies of the NAND cell. The cell moves vertically to align the bus wires, displacement in the x-direction is ignored.

Exiting ICED™

If you are using the demo version, terminate the editor with the QUIT command:

1:**FILE → QUIT**

If you have the full version, exit ICED™ and save your work by selecting:

1:**FILE → LEAVE**

All modified cell files are saved at this time. The LEAVE command saves only cell files whose components have been changed. The EXIT command saves all open cell files whether or not they have changed.

The LEAVE command is generally preferable to the EXIT command, since it leaves the date stamp of unmodified cells unchanged. However, if you edit a cell and change only environment settings (e.g. display or layer parameters), but change no geometry, those new environment settings are lost when you close the editor with the LEAVE command.

The Startup Command File

Back on page 8, you learned that when the layout editor opens to create a new cell, it executes a startup command file. This startup command file is a command file like ED.CMD. It contains a list of layout editor commands.

You can copy the sample startup command file to a new location and edit the commands in it to customize options like colors, layer names, grids, etc. This operation is beyond the scope of this tutorial, but we explore this subject in the next tutorial.

The specification for the startup command file is defined with the START option in the command line stored in the ICWIN.BAT file. The specification in ICWIN.BAT looks like:

START=Q:\ICWIN[7]\TECH\SAMPLES\NEW.CMD

The contents of this command file are shown below.

```
VIEW OFF
$ MENU "M1";  KEEP_LIBRARY_CELLS=ASK;  DISPLAY CELL_DEPTH=100;
PATTERN "SAMPLE";  FILL MIXED ON
AUTOPAN ON  PIXELS=100  SECONDS=0.5;   ARROW MODE=EDIT
DISPLAY CELL_LABELS=ON OUTLINE_DEPTH=1 EDIT_STACK=OFF CURSOR 1 SNAP=ON
SPACER OFF  SPACE=0.0  TRACK_LAYERS=OFF  STYLE=1
VIEW LIMIT ON   SCALE=0.500 DEPTH=1 DOTS=0 UNITS=0.0 SHOW_LAYERS 1:100
ARRAY  DRAW_MODE=SIDES  CELL_LIMIT=1024
TEXT LOWER_CASE=DISABLED MULTI_LINE_TEXT=DISABLED ORIENT=2  DISPLAY_ORIGINS=OFF
USE TEXT_JUSTIFICATION=LB  WIRE_TYPE=2  ARC_TYPE=2  N_SIDES=16
RESOLUTION  STEP=0.500  MODE=SOFT
SNAP  ANGLE=45  STEP=(0.500,0.500)  OFFSET=(0.000,0.000)
NEAR  UNITS=0.05  DOTS=4
```

(Continued on next page.)

[7] Remember that Q:\ICWIN represents the drive and path where you have installed ICED™.

```
COLOR  0 NAME=BLACK        PALETTE=( 0, 0, 0)
COLOR  1 NAME=WHITE        PALETTE=(63,63,63) LEVEL=16
COLOR  2 NAME=YELLOW       PALETTE=(63,63, 0) LEVEL= 6
COLOR  3 NAME=GREEN        PALETTE=(21,63, 0) LEVEL= 6
COLOR  4 NAME=RED          PALETTE=(63, 0,21) LEVEL= 8
COLOR  5 NAME=CYAN         PALETTE=( 0,42,42) LEVEL= 9
COLOR  6 NAME=BLUE         PALETTE=( 0, 0,63) LEVEL=10
COLOR  7 NAME=MAGENTA      PALETTE=(63, 0,63) LEVEL= 8
COLOR  8 NAME=GRAY         PALETTE=(42,42,42) LEVEL=14
COLOR  9 NAME=BROWN        PALETTE=(32,16, 0) LEVEL= 8
COLOR 10 NAME=ORANGE       PALETTE=(63,31, 0) LEVEL= 8
COLOR 11 NAME=PURPLE       PALETTE=(21, 0,14) LEVEL= 3
COLOR 12 NAME=DIM_RED      PALETTE=(22, 0, 0) LEVEL= 3
COLOR 13 NAME=DIM_BLUE     PALETTE=( 0, 0,22) LEVEL= 3
COLOR 14 NAME=DIM_GREEN    PALETTE=( 0,22, 0) LEVEL= 3
COLOR 15 NAME=HI           PALETTE=(63,63,63) LEVEL=15

COLOR BLACK   ONE_DOT=WHITE     FOUR_DOTS=(WHITE,   WHITE,   WHITE,   WHITE)
COLOR WHITE   ONE_DOT=BLACK     FOUR_DOTS=(BLACK,   BLACK,   BLACK,   BLACK)
COLOR YELLOW  ONE_DOT=YELLOW    FOUR_DOTS=(YELLOW,  YELLOW,  YELLOW,  YELLOW)
COLOR GREEN   ONE_DOT=GREEN     FOUR_DOTS=(GREEN,   GREEN,   GREEN,   GREEN)
COLOR RED     ONE_DOT=RED       FOUR_DOTS=(RED,     RED,     RED,     RED)
COLOR CYAN    ONE_DOT=CYAN      FOUR_DOTS=(CYAN,    CYAN,    CYAN,    CYAN)
COLOR BLUE    ONE_DOT=BLUE      FOUR_DOTS=(BLUE,    BLUE,    BLUE,    BLUE)
COLOR MAGENTA ONE_DOT=MAGENTA FOUR_DOTS=(MAGENTA,MAGENTA,MAGENTA,MAGENTA)
COLOR GRAY    ONE_DOT=BLACK     FOUR_DOTS=(BLACK,   WHITE,   WHITE,   WHITE)
COLOR BROWN   ONE_DOT=RED       FOUR_DOTS=(GREEN,   RED,     RED,    YELLOW)
COLOR ORANGE  ONE_DOT=RED       FOUR_DOTS=(RED,     YELLOW,  YELLOW,  RED)
COLOR PURPLE  ONE_DOT=MAGENTA   FOUR_DOTS=(BLUE,    MAGENTA,MAGENTA,BLUE)
COLOR DIM_REDONE_DOT=RED        FOUR_DOTS=(RED,     WHITE,   WHITE,   WHITE)
COLOR DIM_BLUE ONE_DOT=BLUE     FOUR_DOTS=(BLUE,    WHITE,   WHITE,   WHITE)
COLOR DIM_GREEN ONE_DOT=GREEN FOUR_DOTS=(GREEN,   WHITE,   WHITE,   WHITE)
COLOR HI       ONE_DOT=BLACK    FOUR_DOTS=(BLACK,   BLACK,   BLACK,   BLACK)

GRID 1  ON    COLOR=RED     DOTS     STEP=1.0
GRID 2  ON    COLOR=CYAN    CROSSES  STEP=5
GRID 3  OFF   COLOR=WHITE   LINES    STEP=50000

LAYER *  WIDTH=2.0  SPACE=0.0  YELLOW  PAT=0   NO_PEN
INITIALIZE LAYERS 0:255
LAYER 0 PEN=0
LAYER 1 NAME=NWEL WIDTH=3.000 SPACE=0 DIM_BLUE PAT=1 PEN=16 NO_CIF NO_STREAM
LAYER 2 NAME=NDIF WIDTH=3.000 SPACE=0 GREEN    PAT=1 PEN=* NO_CIF NO_STREAM
LAYER 3 NAME=PDIF WIDTH=3.000 SPACE=0 YELLOW   PAT=1 PEN=* NO_CIF NO_STREAM
LAYER 4 NAME=PSEL WIDTH=3.000 SPACE=0 YELLOW   PAT=0 PEN=* NO_CIF NO_STREAM
LAYER 5 NAME=POLY WIDTH=2.000 SPACE=0 RED      PAT=1 PEN=* NO_CIF NO_STREAM
LAYER 6 NAME=M1   WIDTH=3.000 SPACE=0 CYAN     PAT=2 PEN=* NO_CIF NO_STREAM
LAYER 7 NAME=M2   WIDTH=4.000 SPACE=0 BLUE     PAT=3 PEN=* NO_CIF NO_STREAM
LAYER 8 NAME=CONT WIDTH=2.000 SPACE=0 WHITE    PAT=1 PEN=* NO_CIF NO_STREAM
LAYER 9 NAME=VIA  WIDTH=3.000 SPACE=0 GRAY     PAT=1 PEN=* NO_CIF NO_STREAM
GLOBAL KEY.CF9="@UNED"
GLOBAL KEY.F1="RULER"
GLOBAL KEY.F7="DOS"
GLOBAL KEY.F8="@DEEPSHOW"
GLOBAL KEY.F9="DOS"
```

Figure 19: NEW.CMD startup command file

Conclusion

This concludes the Editor Basics tutorial. You now know enough to create cells and modify geometry. Feel free to experiment with the menus to explore the features of the layout editor.

Please read the additional tutorials in the full ICED™ Layout Editor Classroom Tutorials Manual. to help you master other features of the layout editor. These other tutorials are shorter and more focused than this first one. If you take a little more time to learn about advanced features, you will save many hours of work and frustration in the future.

You can see the entire list of tutorials and the subjects they cover on page 5 of this document. You can download the entire manual from the IC Editors website, **www.iceditors.com**.

Index