Senstar 100®

Alarm annunciation and control system

Site Creation installation guide

(Version 5.3)

DA-030207-10, Rev G
Tenth edition
October 2, 2001

Senstar Stellar

Senstar-Stellar Corporation
119 John Cavanaugh Drive
Carp, Ontario
Canada K0A 1L0
Tel: (613)-839-5572
Fax:(613)-839-5830

Website: www.senstarstellar.com
Email address: info@senstarstellar.com

See back cover for regional offices.

DA-030207-10, Rev G
Tenth Edition

Senstar-Stellar Corporation intellectual property is protected by the following patents:

CANADA No.: 1216340, 1332185
U.S. No.: 4562428, 4987394, 5247270
U.K. No.: 2120823, 2163580, 2165681

Patents pending in many other countries.

Table of contents

Drawing maps

Creating datalog and remote alarms

Assigning input and output points

Translating the Site Creation text

Verifying site data

1 Using the Site Creation menus

Overview

This chapter explains:

- how to get into the Site Creation menus
- how to access the primary and secondary monitors
- how to use the mouse and the keyboard to move around in the menus
- how to access the fields in the menus

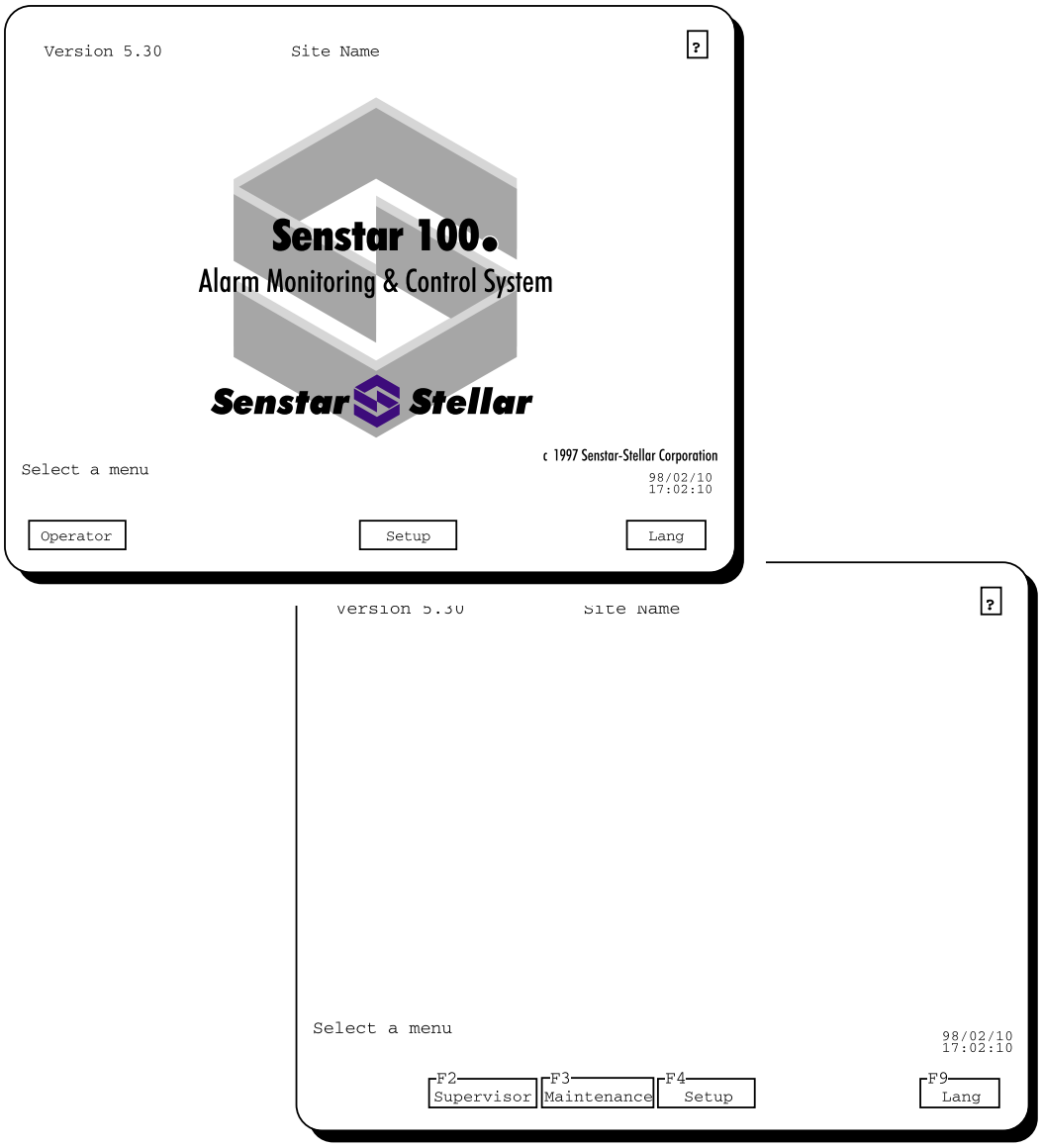
The Site Creation software should be installed when you receive your system. If it is not, or if you are upgrading the software, refer to Appendix A for installation instructions. It is recommended that you read this chapter first to familiarize yourself with using the Site Creation package.

Using two monitors

In order to run the Senstar 100 Site Creation software, you need to have both the primary and the secondary monitors. You use the secondary monitor to display the menus, enter text information and perform most of the commands and functions. You use the primary monitor to draw and display your site maps.

Entering the Setup menu

The Senstar 100 top menu allows access into Senstar 100's four system menus — Operator, Supervisor, Maintenance and Setup. The top image appears on the primary screen and the lower image on the secondary screen as shown below.



If the system is configured for two-language operation, the **Lang** function also appears.

Setup

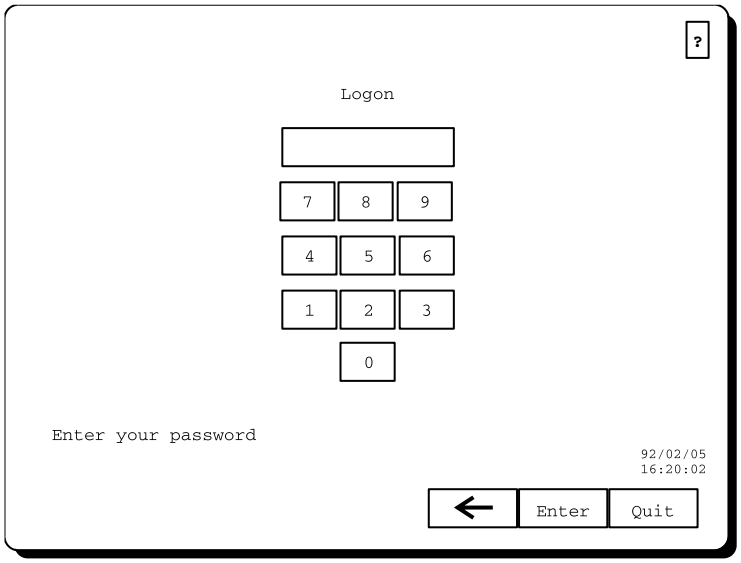
The Setup function appears on both the primary and the secondary screens. The Site Creation software is accessed through the Setup function on the secondary screen. Choose the **Setup** function on the secondary screen by pressing the **F4** function key on the keyboard.

Passwords

You might want to use passwords for security reasons. Passwords are assigned in the Supervisor menu (see the ***Senstar 100 Supervisor's Guide***, DA-030204).

If you don't require a password, the Setup menu is displayed as soon as you choose the Setup function.

If you require a password to enter the Setup menu, the logon menu is displayed when you select the Setup function.



Each user is assigned a six-digit password. Enter the password by touching the numbers on the displayed keypad, or by pressing the numbers on the keyboard. The numbers don't appear on the screen; instead, you see a small box appear for each digit you type. This feature provides added security.

If you typed a digit incorrectly, use the **F8** (←) function to delete it. Then re-enter the correct digit.

Press the **F9 (Enter)** function once you've entered your password correctly.

Incorrect passwords and duress alarms

If the password that you enter is incorrect, the system returns to the menu selection display. You must select the Setup menu again, and re-enter your password.

If you have configured your system for duress alarms, a console tamper alarm will activate if you make more than three failed attempts to enter your password correctly. The alarm is generated at a remote location. This configuration ensures that unauthorized users can't repeatedly try to type in passwords and gain entry to the system.

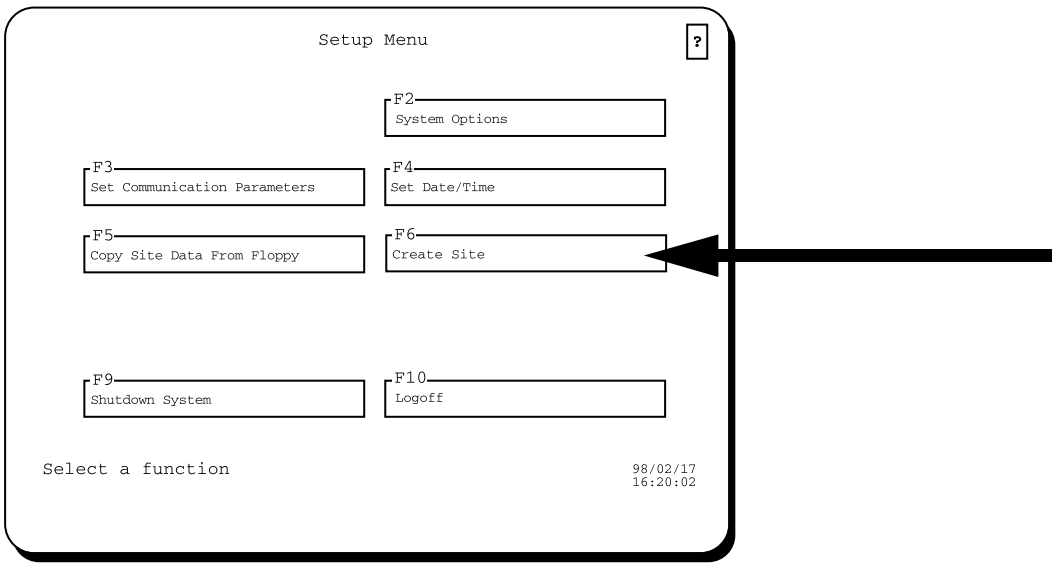
An alarm will also be activated at a remote location if you enter a duress code when entering the password. The duress code is a digit that alerts the system about an operator problem, while still allowing access to the system menus. To generate a duress alarm, enter your password, but replace the last digit with your duress code.

When you've entered your password correctly, the secondary screen changes to display the Setup menu. The primary screen doesn't change.

Entering the Site Creation menu

When you access the Setup menu, you'll see the Setup functions displayed on the secondary screen. The Site Creation software is accessed through function **F6 - Create Site**.

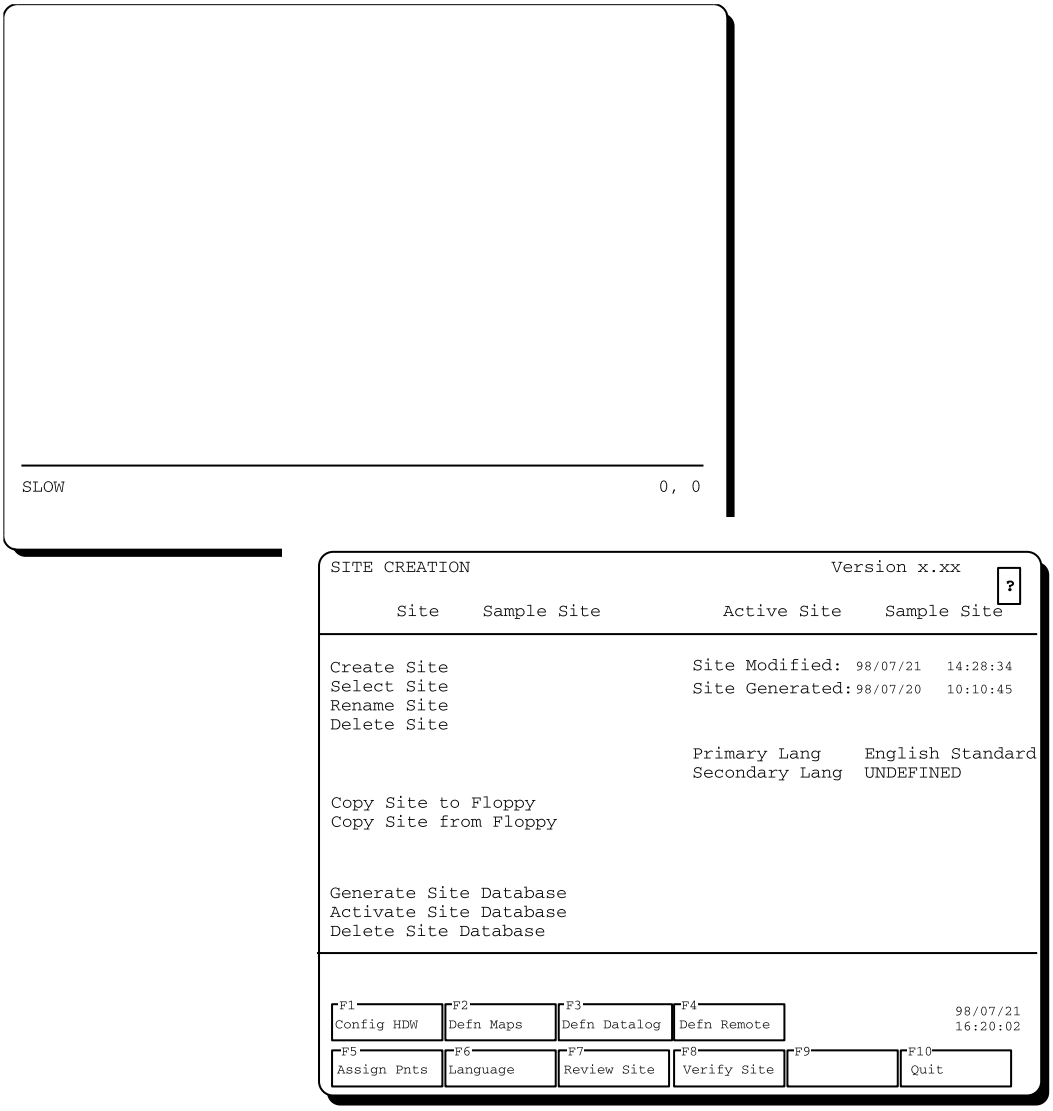
Press **F6** to enter the Site Creation menus.



You'll be warned that alarm processing capabilities will be disabled if you continue. Press **F1** to continue, or **F2** to cancel and return to the Setup menu.

If alarm processing is in progress, you won't be able to enter Site Creation. When you press F1 to continue, you'll get an error message.

If you're allowed access to Site Creation, the Site Creation main menu appears on the secondary screen.

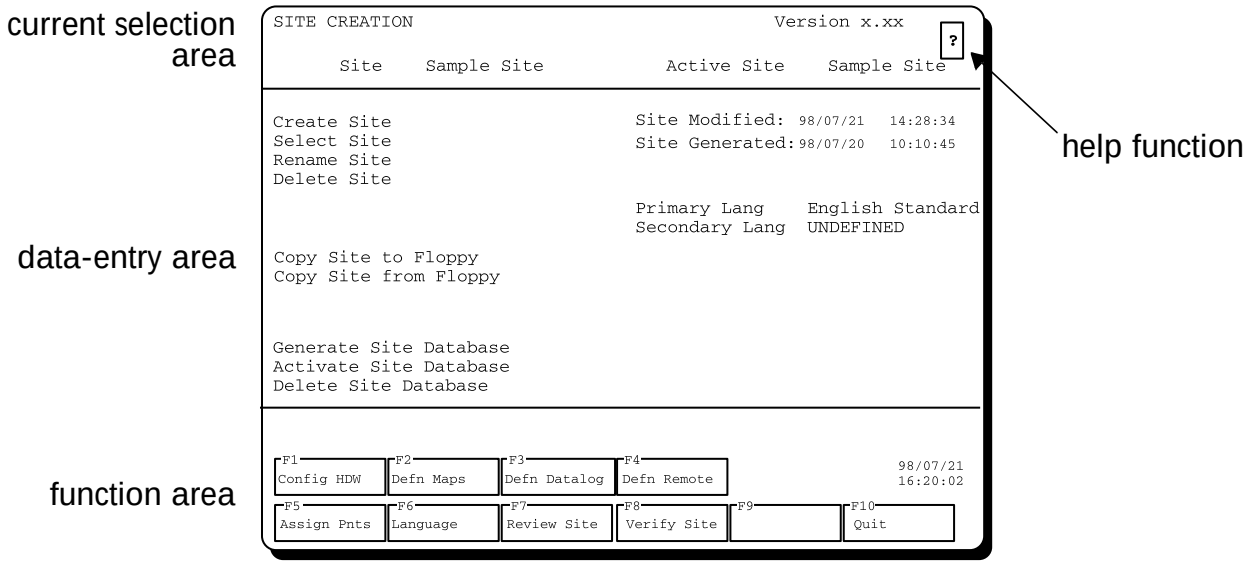


The secondary monitor displays the Site Creation main menu. In this menu you manage up to 8 site databases into which you'll put all the information you've collected about your site(s). You then access each of the functions in the function key section of the menu to enter your site data.

The primary monitor displays a nearly-blank screen and a status line. Maps are drawn and displayed on this screen.

Secondary screen

The secondary screen is divided into three main areas.



Current-selection area

The title of the menu appears in this area. Any applicable site, map or device names also appear here. The fields may be blank until you select a name in the middle area of the menu. You can't select or change any of the fields in this area.

Help function (?)

The **Help** (?) function also appears in the current-selection area, and it is the only function in this area that can be accessed. When you select Help, you get information on the functions currently available to you.

Data-entry area

In this area, you fill in data relating to a particular site, map or device whose name is displayed in the current-selection area of the menu. You also use the data-entry area to execute general commands for copying, saving or deleting data, and to execute specific commands for designing site maps.

You can move around in the data-entry area by using the mouse or the cursor keys. You then use the mouse or the keyboard to input information (See ***Using the mouse or the keyboard***, page 1-11)

The data-entry area contains the following menu fields:

Field name	Purpose
Status field	displays a name or value that has been defined by another field on the menu, from another menu, or by the system (e.g., "Site Created" field)
Input field	opens up a field where you enter a new name or value via the keyboard
Pop-up menu	displays a list of pre-defined or user-defined names or values from which you select what you want
Command	when selected, performs a particular function (e.g., saving or deleting your work)

The above fields will be discussed in more detail in, ***Input fields, pop-up menus and commands***, page 1-14.

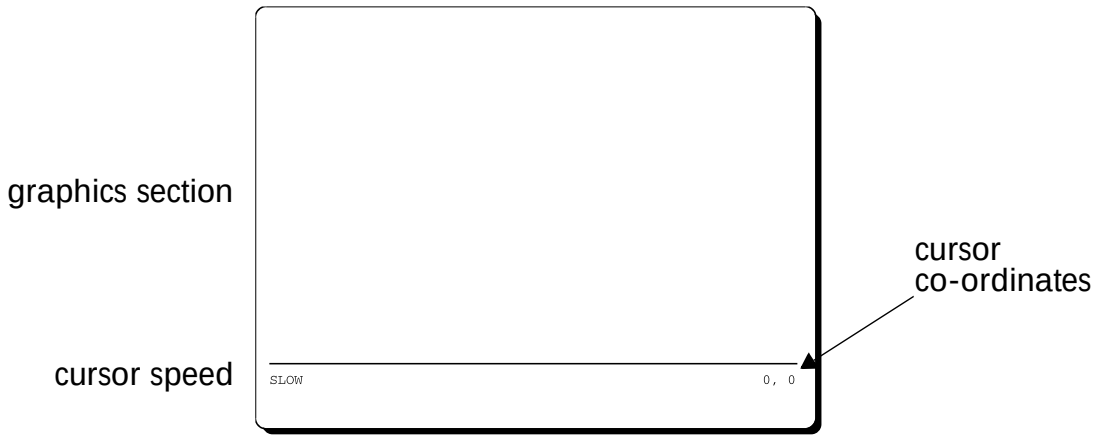
Function area

The function keys are located in this area. You use the function keys to move into submenus, or to scroll through lists of sites, maps or devices. You select a function key by using the keyboard or the mouse.

The prompt line, any error messages, and the time/date display also appear in this area.

Primary screen

When you first access the software, the primary screen appears as shown below.



The largest area of the screen is the graphics section. This is where you draw your maps by using the mouse or keypad, and where the maps are displayed. Whenever you perform a function on the secondary screen that involves a map, the map is automatically displayed in the graphics section of the primary monitor.

The bottom area of the screen is the status area. It indicates the speed of the cursor when you're drawing maps (fast for regular drawing, slow for fine detail), and where the cursor is located. Both settings are made from the secondary monitor; you can't change them on the primary screen.

Moving between screens

You can only access the primary screen when you're drawing maps. To move control from the secondary screen to the primary screen, you must select one of the drawing commands (e.g., Draw Line, Draw Zone).

To move control from the primary screen back to the secondary screen, press the **F10** function key.

Using the mouse or the keyboard

You can use either the mouse or the keyboard to move the cursor and access the functions on the screens. Remember that you can only access the primary screen when you're drawing maps.

Moving the cursor with the mouse

To move your cursor around on the secondary screen, simply move the mouse until the cursor is positioned where you want it. Whenever the cursor is on a field that can be selected, that field becomes highlighted. When it is on an unselectable field, such as a status field, the cursor appears as an underscore.

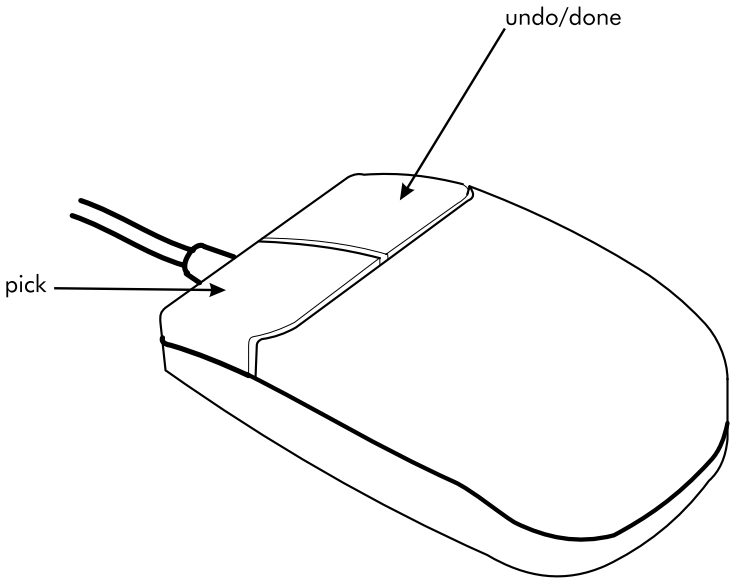
You can highlight:

- all the selectable fields in the data-entry area of the screen (i.e., input fields, pop-up menus and commands)
- the help function in the current-selection area
- the function keys in the function area

To move your cursor around on the primary screen, move the mouse. The cursor appears as a white cross.

How to select fields and functions

Use the two buttons on the mouse to select fields and functions. The diagram that follows shows how the buttons have been configured.



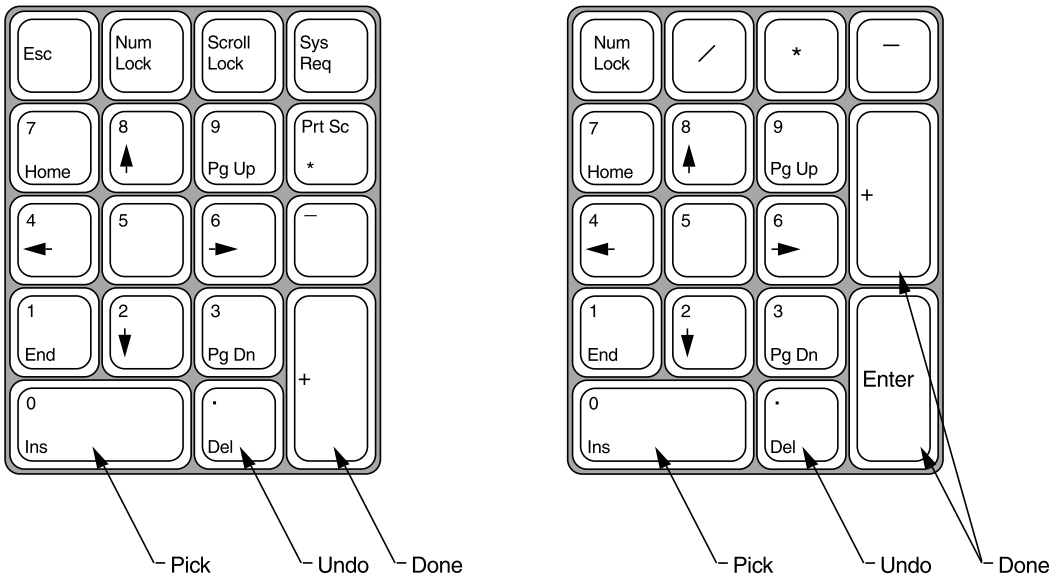
On the secondary screen, the **pick** button is used to select a function or field after you've highlighted it with the cursor. On the primary screen, the **pick** button is used to select the point where you want to start drawing an object.

On the primary screen, the **undo/done** button is used to cancel a point, i.e., undo undoes your last **pick**.

On the primary screen, the **undo/done** button completes a function. For example, when you're drawing objects on the primary screen, you press the **undo/done** button to indicate that you've finished drawing.

Moving the cursor with the keyboard

You can use the keyboard instead of the mouse to move the cursor around on the screens and select functions. You perform most of the functions by using the keypad (located on the right side of the keyboard). Two keypad configurations are shown below.



The **Ins**, **Del**, and + and **Enter** keys are configured as the **pick**, **undo** and **done** functions. They work in exactly the same way as the mouse buttons.

On the secondary screen, the ↑ and ↓ keys move up and down columns, while the → and ← keys move between columns. When the cursor is on a field, the field is highlighted. If you try to pick an unselectable field, you'll get an error message.

On the primary screen, the cursor appears as a white cross. Use the ↑, ↓, →, ← and diagonal keys to move the cursor around on the primary screen.

The cursor keys on your keypad only let you move around in the data-entry area of the secondary screen. You have to use other keyboard keys to perform other functions. To access the help function, press **Shift-?** on the keyboard. To access the function keys, press the function keys on the keyboard.

Input fields, pop-up menus and commands

There are three ways to input or retrieve information in the data-entry area of the secondary screen:

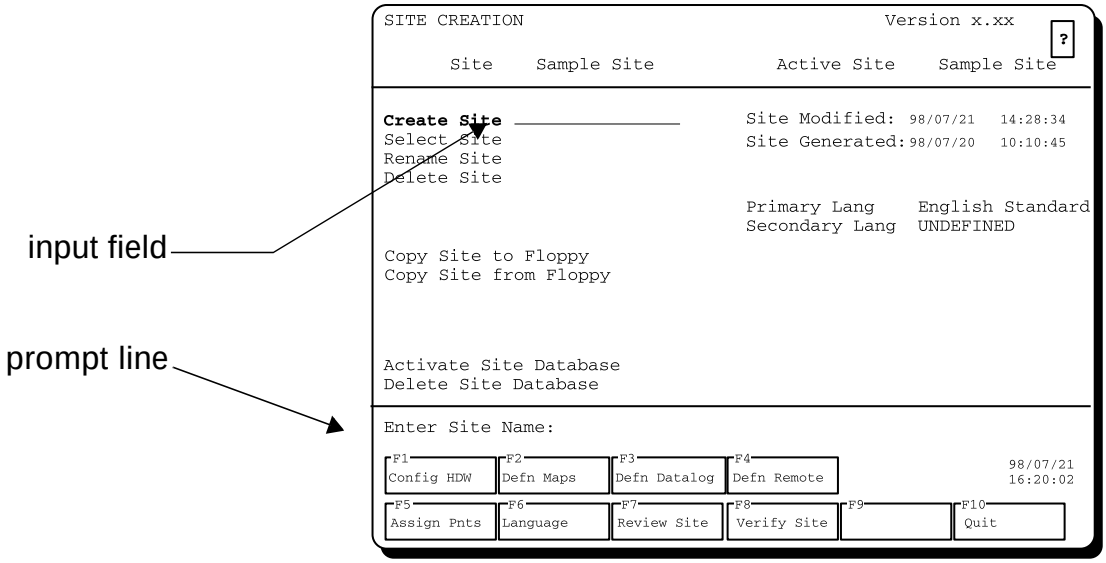
- input fields
- pop-up menus
- commands

Input fields

An input field is a field that you enter information into using the keyboard. Use the input fields to enter the name of a site, map, zone, etc.

To select an input field:

1. Move the cursor to highlight the field.
2. Press the **pick** key (leftmost button on the mouse, or the **Ins** key on the keyboard).
3. The field opens up and you're prompted to enter a name.



Use the keyboard to perform the following functions in an input field:

Function	Keyboard command
Enter text	use the keyboard to enter a name
Save text	press the ↵ **Enter** key to save the text after you've keyed it in. The field closes up and the name is saved.
Cancel text	press the ↵ **Enter** key before keying in a name to cancel the function. The field closes up.
Insert text	press the **Ins** key to insert text before the cursor **Note**: This applies only to input fields longer than 5 characters.
Delete text	press the **Del** key to delete the current character in the field. Press the ← (backspace) key to delete the previous character in the field.
Move within the field	press the ← and → keys to move along the text in the field

If you have two or more input fields side-by-side, you can also do the following, for example when entering a device, card and point:

Function	Keyboard command
Move to the next field	press the **Tab** key or the ↓ key to move to the next input field
Move to the previous field	press the **Shift-Tab** key or the ↑ key to move to the previous input field
Move to the last field	press the **End** key to move to the first character in the last input field in the line
Move to the first field	press the **Home** key to move to the first character in the first input field in the line

Entering special characters in an input field

To enter a **?** in an input field, hold down the **Alt** and **Shift** keys and press ?.

If you have the Standard English keyboard, but wish to use special characters such as accents (for example, you want to enter French text using the Standard English keyboard), use the **Alt** key and the character codes as shown below.

Code	Character	Code	Character	Code	Character	Code	Character
Alt C ,	Ç	Alt E ´	É	Alt N ~	Ñ	Alt g d	δ
Alt u "	ü	Alt A e	æ	Alt a _	ª	Alt o /	∅
Alt e ´	é	Alt A E	Æ	Alt o _	º	Alt F1	⌜
Alt a ^	â	Alt o ^	ô	Alt Shift?	?	Alt F2	⌝
Alt a "	ä	Alt o "	ö	Alt o o	∞	Alt F3	⌞
Alt a ’	à	Alt o ‘	Ò	Alt g a	∝	Alt F4	⌟
Alt a o	å	Alt u ^	û	Alt g b	ß	Alt F5	⌠
Alt c ,	ç	Alt u ‘	Ù	Alt g g	Γ	Alt F6	⌡
Alt e ^	ê	Alt y "	ÿ	Alt p i	π	Alt F7	+
Alt e "	ë	Alt O "	Ö	Alt g e	Σ	Alt F9	├
Alt e ’	è	Alt U "	Ü	Alt g s	σ	Alt F10	┤
Alt i "	Ï	Alt a ´	á	Alt g u	µ	Alt F11	┬
Alt i ^	î	Alt i ´	Í	Alt g t	τ	Alt F12	┴
Alt i ‘	Ì	Alt o ´	Ó	Alt o |	ϕ		
Alt A "	Ä	Alt u ´	Ú	Alt o -	θ		
Alt A o	Å	Alt n ~	ñ	Alt g w	Ω		

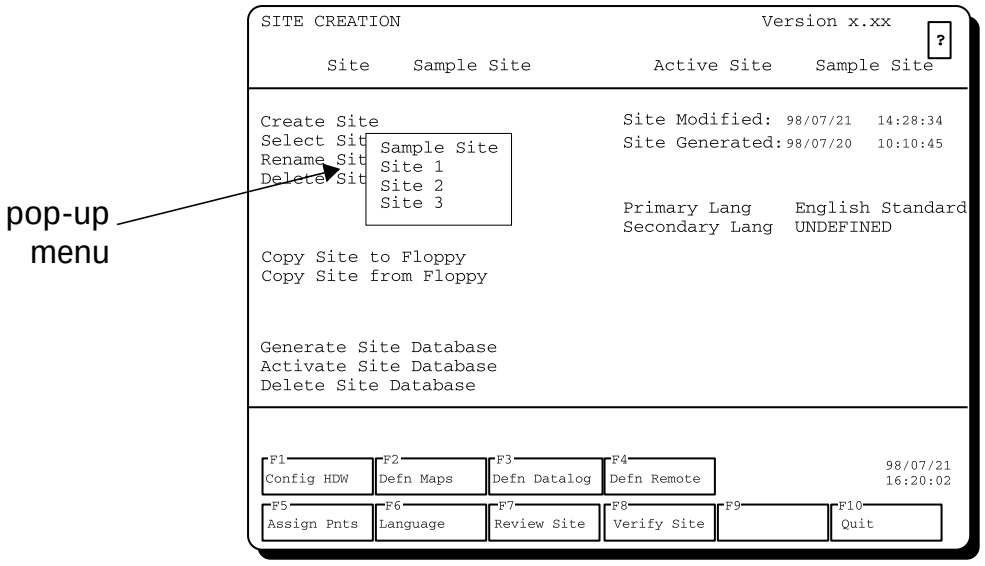
To enter a special character, press and release the **Alt** key, then press the corresponding character keys.

Pop-up menus

Pop-up menus appear when you have a number of options with the function.

To select a function with a pop-up menu:

1. Move the cursor to highlight the function.
2. Press the **pick** key to select the function. The pop-up menu appears, containing a list of options.
3. Scroll through the options by using the cursor keys or the mouse. Highlight the option you want, and press the **pick** key to select it.



To cancel the function, press the **undo** key after opening up the pop-up menu. The menu disappears.

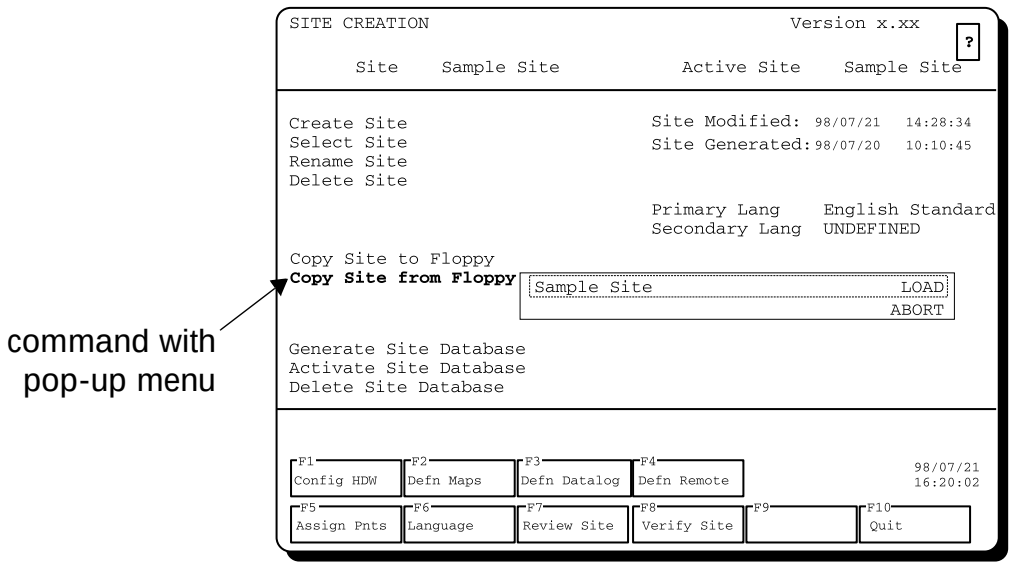
Commands

Commands are used to perform tasks such as copying, deleting or saving data. You also use commands to draw your site maps.

To execute a command:

1. Highlight the function with the cursor keys or the mouse.
2. Press the **pick** key to select the function. A pop-up menu **may** appear, asking you to verify that you want to continue.You have the choice of continuing or cancelling the command.

When the command is initiated, a prompt or message appears on the prompt line (function area of the secondary screen), informing you of the command in progress. Do not try to cancel the command at this point.



2 Creating a site creation database

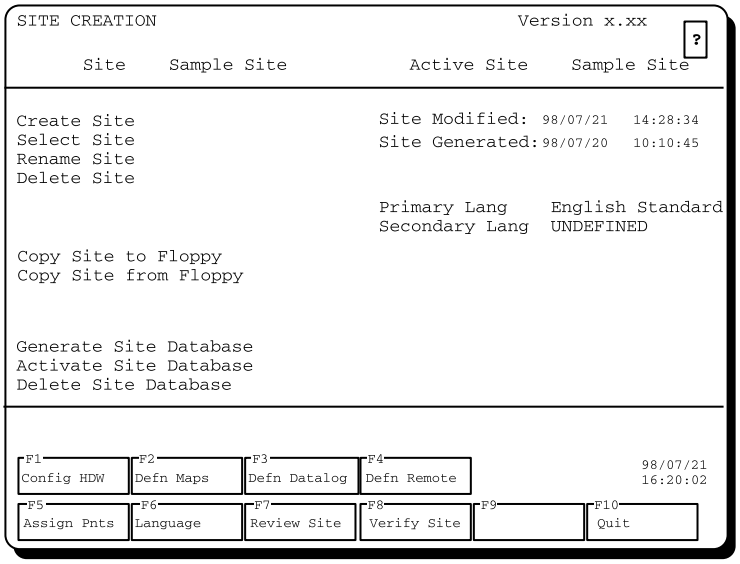
Overview

The first step in entering the site data is to create a site creation database that will hold all the information for your site.

The chapter explains:

- how to create a new site creatioin database for your site
- how to save, back up and restore any site data
- how to use the function keys to access all the menus you'll need

Creating a new site creation database



When you access the Site Creation menu, the main menu commands and functions are displayed on the secondary monitor.

Site field

The **Site** field displays the name of the current site. If the sample site has been installed, its name appears here; if not, the field is blank.

> If the sample site is not installed and you want to install it, refer to Appendix A.

Active Site field

The **Active Site** field holds the name of the site used by the Senstar 100 application system when you boot up.

The **Create Site** function is highlighted when you enter the main menu. To create a new site creation database:

1. Select the **Create Site** function by pressing the **pick** key. An input field opens up and you're prompted to enter the name of your site.
2. Key in the name of your site (e.g., Demo Site). You can have up to 15 characters in your site name, including spaces.
3. Press the **Enter** key to finish the function.

The system now creates an empty site database with the name that you've given it. The new site name appears in the Site field, in the current-selection area of the menu.

If you key in an existing site name, you'll get an error message. You will have to enter another name.

If you wish to change or delete the current site, see ***Editing an existing site***, page 2-4. Otherwise, go to ***Using the function keys***, page 2-6.

You can create a maximum of eight sites.

Editing an existing site

The following functions can be used to make modifications to an existing site.

Selecting another site

To select another site that you want to work with:

1. Choose the **Select Site** function. A pop-up menu appears that contains a list of all the sites that have been created.
2. Scroll through the list and highlight the one you want, then press the **pick** key to select it. The pop-up menu disappears and the site name appears in the Site field (current-selection area of the menu). This site becomes the current site. Any modifications you make will affect this site.

If no sites have been created, you'll get an error message when you pick this function.

Renaming a site

To change the name of the current site:

1. Choose the **Rename Site** function. An input field opens up, and you're prompted to enter the new site name.
2. Key in the revised name in the input field.
3. Press the **Enter** key to finish the function. The revised site name appears in the Site field.

If you rename the site to an existing name, you'll get an error message.

Deleting a site

To delete the current site:

1. Choose the **Delete Site** function. A pop-up menu appears, confirming that you want to delete the site. Keep in mind that if you delete the site, you also delete any data that you may have entered in the database.
2. Pick **Yes** to delete the site, or **No** to cancel the function. You can also press the **undo** key to cancel the function.

Changing the language of a site

The default primary language of the site is English Standard. You can set the primary site language to any other language that has been created (see chapter 7, ***Translating the Site Creation tex***t). When the secondary language is undefined, the system operates only in the primary language. You can set the system to operate in two languages by setting the secondary language to any other language that has been created.

Using the function keys

Once you've created a new site, you use the function keys to go through the procedures of entering data in the database.

The following table lists the function keys available from the main menu. Chapters 3 to 10 explain each of these functions.

Function key	Description
F1 - Config HDW	enter the hardware configuration, including location of the ports, number of primary and secondary devices, and configuration of the primary and secondary devices
F2 - Defn Maps	draw your site maps on the computer, and specify zones, groups, sensors and camera locations
F3 - Defn Datalog	create datalog alarms (alarms that are recorded on the event logger)
F4 - Defn Remote	create remote alarms
F5 - Assign Pnts	specify input and output points for each alarm you've identified in the system
F6 - Language	define and edit the languages for your sites
F7 - Review Site	review hardware configurations and point assignments
F8 - Verify Site	check that point assignments are valid and complete
F10 - Quit	exit from the Create Site function and return to the Setup menu.

If your system has two keyboard character types (e.g., English and Arabic) defined, the **F12** function key can be used to toggle between the two character sets whenever you're entering text. The F12 key is not shown on the menu.

Backing up site creation data

Every time you add data to your site, you should make a backup on floppy disks. To back up your site data:

1. Select the **Copy Site to Floppy** function on the main menu. A pop-up menu appears, confirming that you want to back up your site.
2. Insert a floppy disk and pick **Backup** from the pop-up menu.

To cancel the function, pick **Abort** or press the **undo** key.

The backup procedure formats the floppy disks; therefore, any data on the disks will be erased.

If you're backing up more than one site, use one set of disks per site.

The backup procedure creates one disk (or more, if the site is large) for the site creation database information. If you back up the database after generating it with the Generate Site Database command, you will also need a disk(s) for the (run-time) site data (see chapter 9, Generating a site database).

Recovering corrupted data

If the database on the hard disk is ever corrupted (due to a power failure, system failure, etc.), you can restore it from your backup floppy disks. To restore the data:

1. Select the **Copy Site from Floppy** function. A pop-up menu appears, confirming that you want to restore your data.

> **CAUTION** Make sure that you have selected the appropriate site by choosing it from the Select Site pop-up menu, ***before*** choosing this function. This site is the one that you'll load your data into.

2. Insert the backup Site Creation data disk and pick **Load**.

To cancel the function, pick **Abort** or press the **undo** key.

This function restores the Site Creation databases only. To restore the active (run-time) site data, you must load the site data disks from the Startup/Setup menus, or generate and activate a new (run-time) site database.

> **CAUTION** Loading the (run-time) site data from the Startup/Setup menus loads the active site only.

See the Senstar 100 Installation Guide (DA-030202), for instructions on loading the site data. See chapter 9, Generating a site database, for information on generating and backing up a new database.

3 Entering hardware configuration data

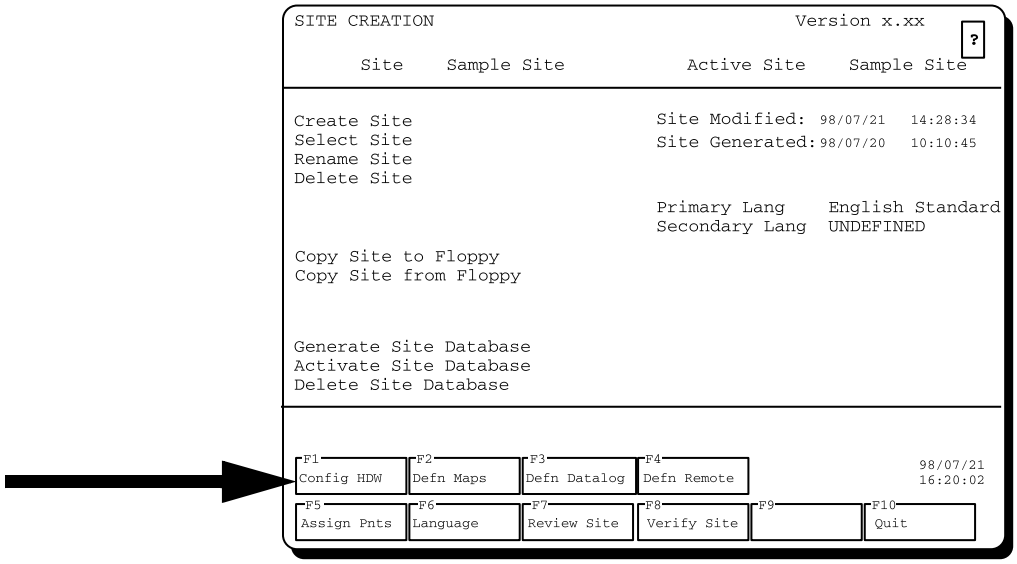
Overview

Once your database has been created, you must enter the hardware configuration of your system, including the location of ports in the system unit, the types of primary and secondary devices, and the configuration of each primary and secondary device in the system.

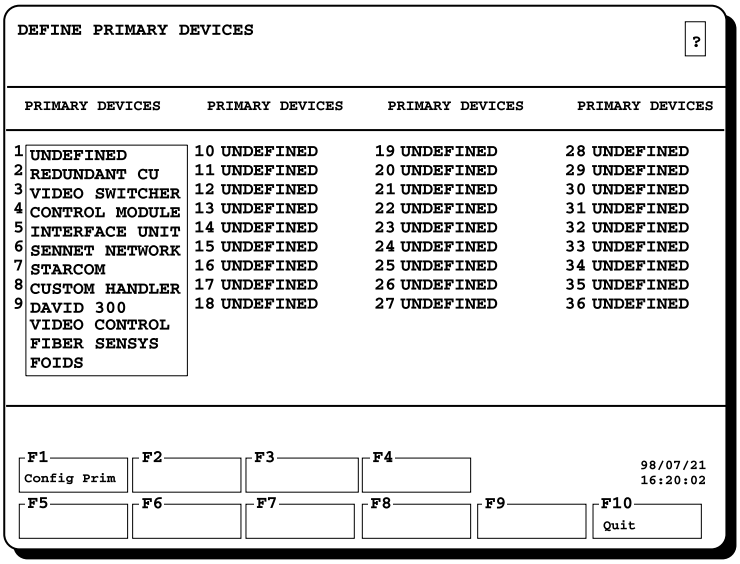
This chapter explains how to enter the hardware configuration data in the site creation database.

Defining the primary devices

To enter the hardware configuration data, choose **F1 - Config HDW**, from the main menu.



The screen displays the Define primary devices menu. Primary devices are devices that collect or respond to alarms.You must define all the primary devices in your system.



1. Select **1**. A pop-up menu appears, containing the possible primary devices.
2. Select a device from the list.
3. Select **2** and repeat the above steps. Continue this procedure until you have listed all the devices in your system.

> Move through the devices sequentially. Do not leave a device unselected if you're going to select the next one - i.e., don't leave device 1 unselected if you're going to select device 2.

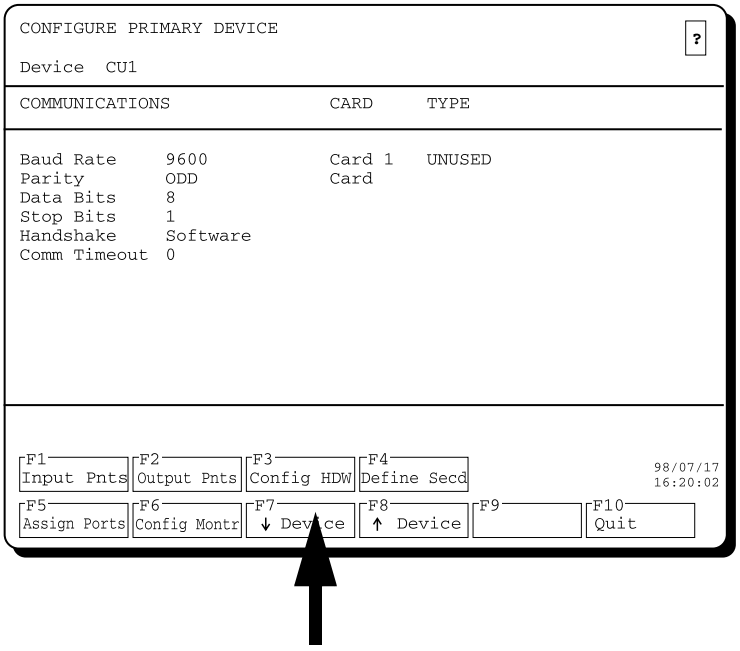
If you have more than one of the same device, you must specify it twice. For example, if you have two Control Modules, select Control Module for device 1 and for device 2. The Control Modules will automatically be assigned as CM1 and CM2.

The Redundant CU in this list refers to the control unit for a redundant system.

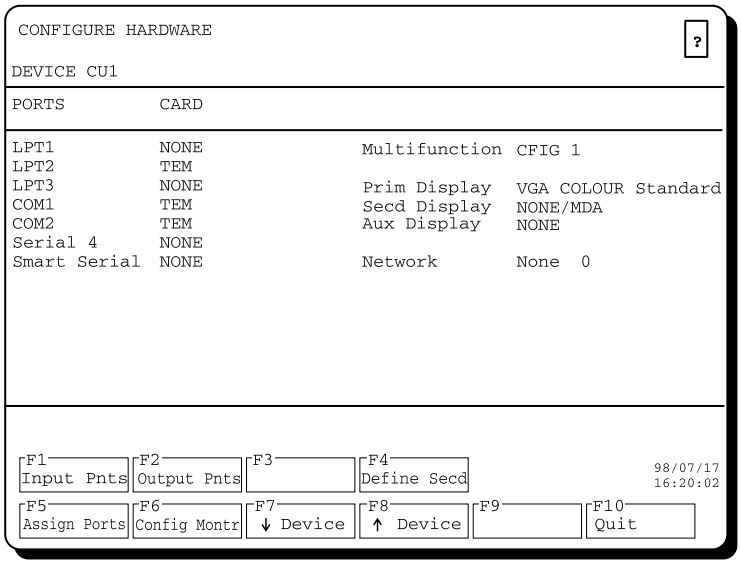
Defining hardware devices

When entering hardware devices, keep in mind that you are entering the configuration for the target system (the system that will run Senstar 100). The target system might not be the same as the system you are working on.

To configure the hardware devices press **F3 - Config Hdw** from the Configure primary devices menu.



The screen displays the Configure Hardware menu.



Listing ports and cards

The **Ports** column lists the possible ports in your system unit. To enter the ports and cards:

1. Select each port that is in your system unit. When you select a port, a pop-up menu appears, containing a list of card options.
2. Choose the option that is applicable for the port.
3. Select the **Multifunction** card and pick the appropriate configuration from the pop-up menu.
4. Select the **Prim Display** function and pick the appropriate primary display monitor from the pop-up menu. The monitor type is the one used by the target system; this might not be the same as the type you're using now.
5. Select the **Secd Display** function and pick the appropriate secondary display monitor from the pop-up menu.
6. Select the **Aux Display** function and pick the appropriate auxiliary display monitor (if applicable) from the pop-up menu.
7. Select the **Network** function and pick the appropriate network type from the pop-up menu.

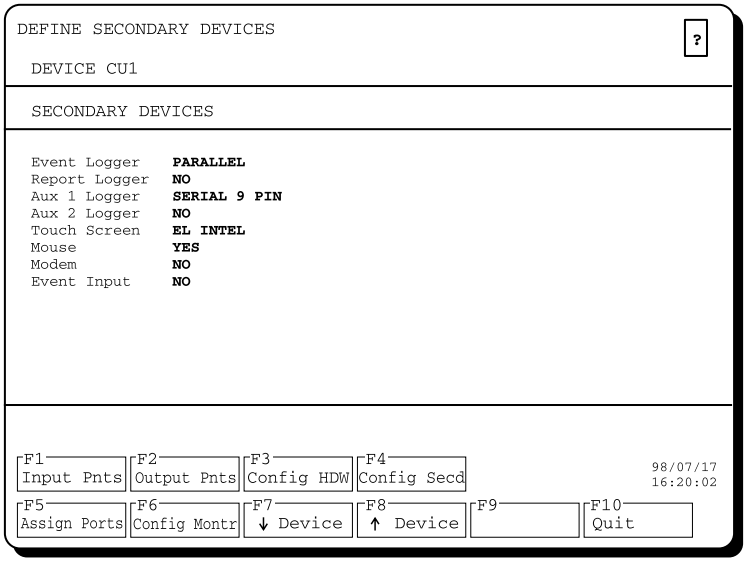
Refer to the Senstar 100 Installation Guide (DA-030202) for more information on choosing ports and cards.

Defining secondary devices

Secondary devices are devices such as printers, touch screen and mouse. To define the secondary devices in your system:

To define the secondary devices press **F4 - Defn Secd** from the Configure primary device.

The screen displays the Define secondary devices menu.



1. Scroll through the list of devices, and select each one that is present in your system.
2. Pick **Yes** from the pop-up menu to include the device, or specify the device type, if applicable.

It's assumed that you always have an event logger. Specify whether it's serial or parallel, 9 pin or 24 pin.

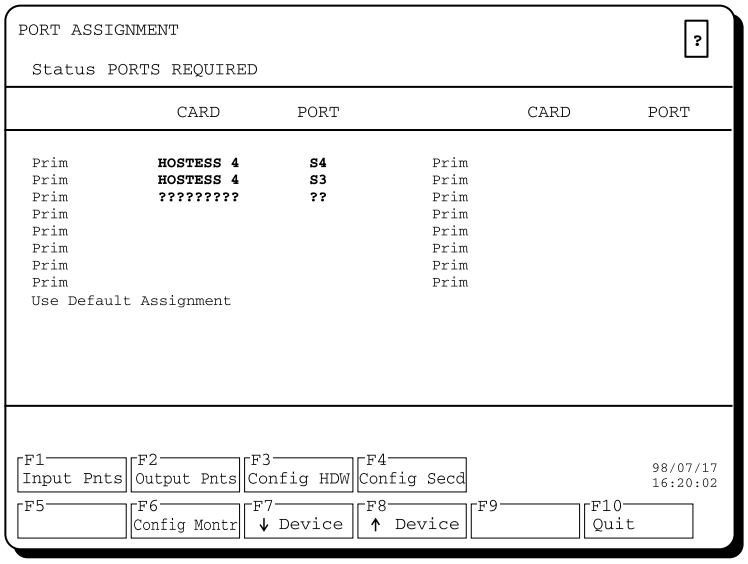
The pop-up menu for the touch screen lists the touch screens currently supported by Senstar-Stellar. If your touch screen is not in the list, contact Senstar-Stellar for details on how to configure the touch screen.

Port assignments

When you have entered all the data for the hardware configuration ports and cards of all CU's and secondary devices, ensure that you have specified the correct number of ports for the number of devices.

Press **F5 - Assign Ports**, from the Configure primary devices menu.

A sample Port Assignments screen is shown below.



The status line indicates whether all devices have been assigned to ports; if yes, the status is **Success**.

If you don't have enough ports for all the devices specified, the status field indicates **Ports Required**. The unassigned devices have question marks in the card and port columns.

Select **Use Default Assignment** to enable the system to select the ports for you. Change the port assignments as required using the pop-up menus.

If you get the Ports Required message, return to the Configure Hardware menu by pressing **F3**, and check that you have entered the correct data for the ports. Also check the lists of primary and secondary devices. Make any required changes, and select the Assign Ports function again.

WARNING

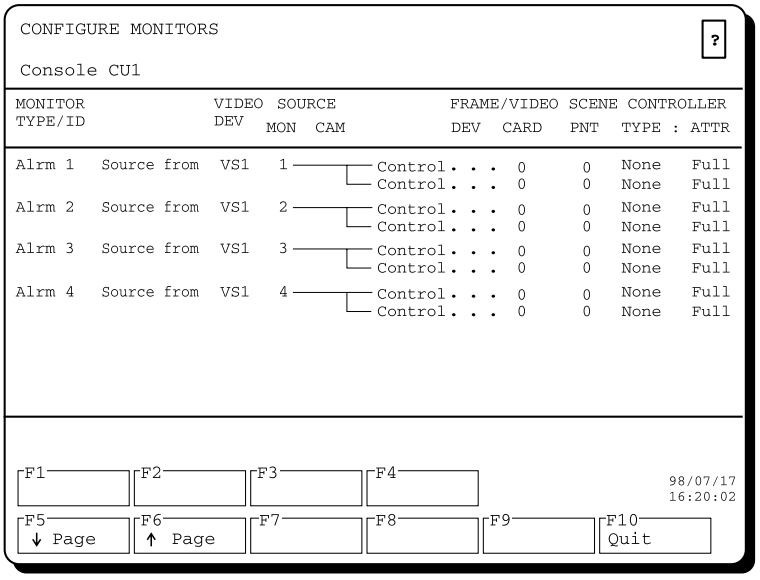
You must connect your devices to the assigned ports exactly as specified on the Port Assignments screen. If you don't, serious system problems could occur.

Configuring system monitors

System monitors are configured through channels. You must specify the video source and frame/video scene controller for each monitor channel that is being used. For each CU, 8 alarm monitor channels are available for the cameras that are assigned to display zones, and 4 manual monitor channels are available for the individual operator selected cameras.

To enter the control unit monitor data select **F6 - Config Montr** from the Configure primary devices menu.

The screen displays the Configure monitors menu. A typical configuration is shown below.



Each monitor channel can have up to 2 video controllers connected to it that will provide frame grabbing or quad video features, and each channel can fan out to more than one physical video monitor.

Configuring primary devices

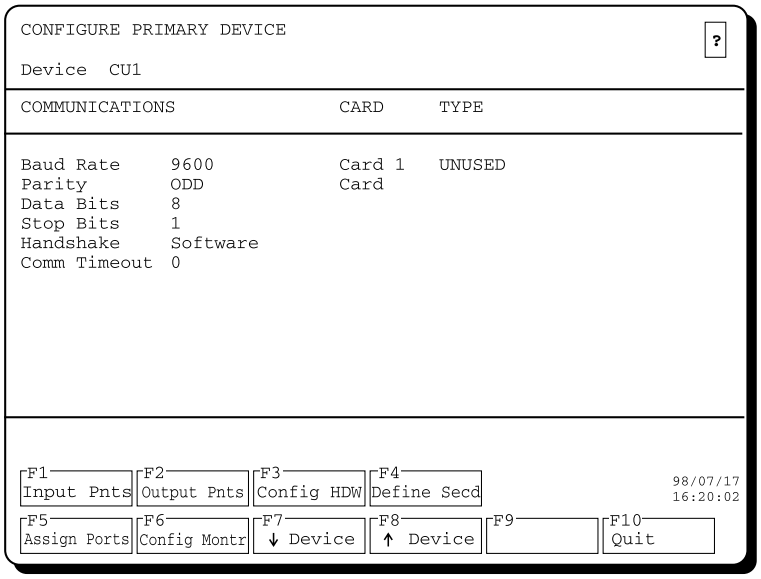
For each primary device that you've listed, you must specify the communications parameters (baud rate, parity, data bits, stop bits, handshake and timeout), the cards associated with the device, and the normal state of any input and output lines.

To enter the data, press **F1 - Config Prim**, from the Define Primary Devices menu.

The screen displays the CU1 device. The device is identified by its label. Move through the other primary devices with the **F7 (↓ Device)** and **F8 (↑ Device)** scrolling keys.

Entering data on a control unit

The menu for the control unit is shown below.



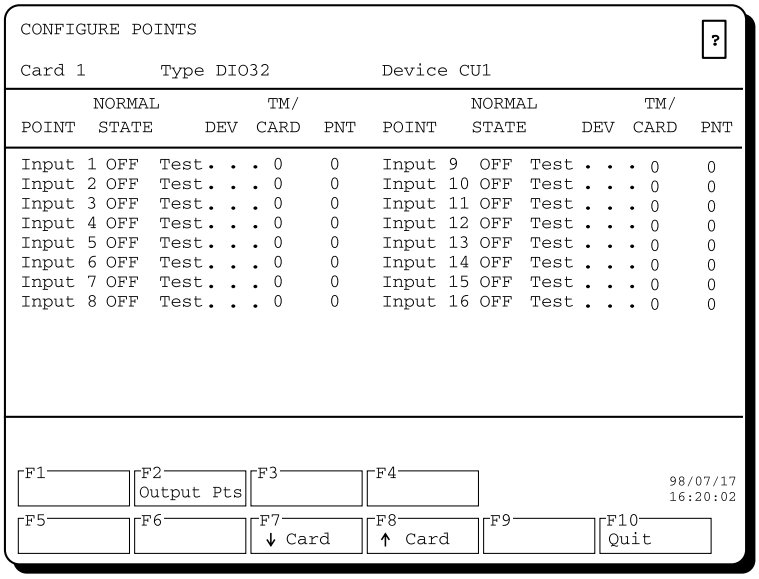
The default parameters are displayed. You can't change the Data Bits field. Select the other settings from the pop-up menus.

To specify a timeout, enter a number from 1 to 60 seconds in the input field. For no timeout, enter 0.

Configuring input points for a Control Unit

If you have input cards installed in the Control Unit, you must specify the normal state and test devices attached to each point on each card.

Select **F1 - Input Pnts**.



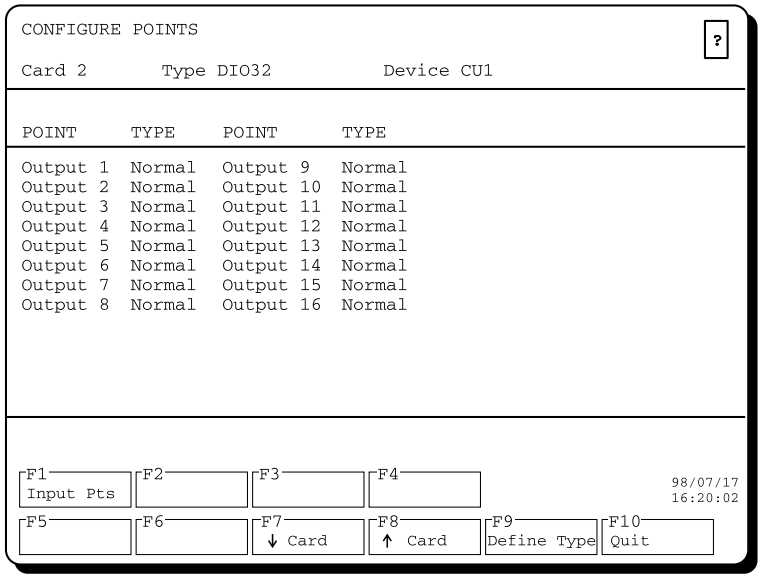
To change the default normal state for an input point, select the point and pick On from the pop-up menu.

To enter a test device for an input, select the **Test** field. Input fields appear for the test device, TM/card number and point number. In the Dev field, enter the 3-character device code (e.g., IU1). Then enter the TM or card number, and the point number.

Configuring output points for a Control Unit

If you have output cards installed in the Control Unit, you must configure each point on each card.

Select **F2 - Output Pnts**.



The output points are listed with the default type.

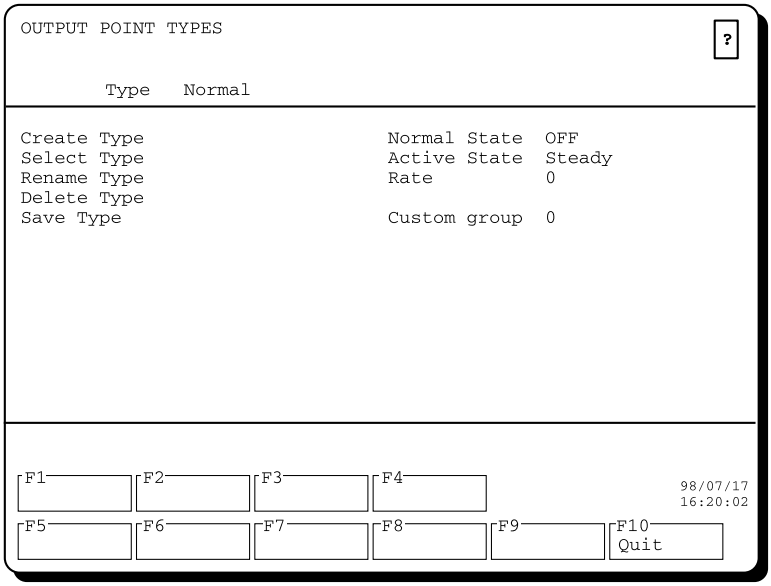
In this menu, you:

- define the output point types
- configure each output point

Output points are defined in terms of normally on/off contacts and the active state (steady, pulse rate or flash rate). An output point type is defined as a combination of these parameters. When you define an output point type, you can then assign a type to each of the output points. This saves you from having to enter each parameter individually for each output point.

Defining output point types

Press **F9 - Define Type** to define the output point types.



The top line displays the output point type that you're defining. In this case, a system default type **Normal** has already been created.

To create a new output point type, enter a type name in the **Create Type** field. Names can be up to eight characters, including spaces. You can create up to eight output point types for the system.

For each output point type, specify:

- normal state — **Off** or **On**
- active state — steady, pulse or flash
- rate — if you select pulse as the active state, enter a value from 500 to 5000 (in increments of 100). If you select flash as the active state, enter a value from 1000 to 5000 (in increments of 100)

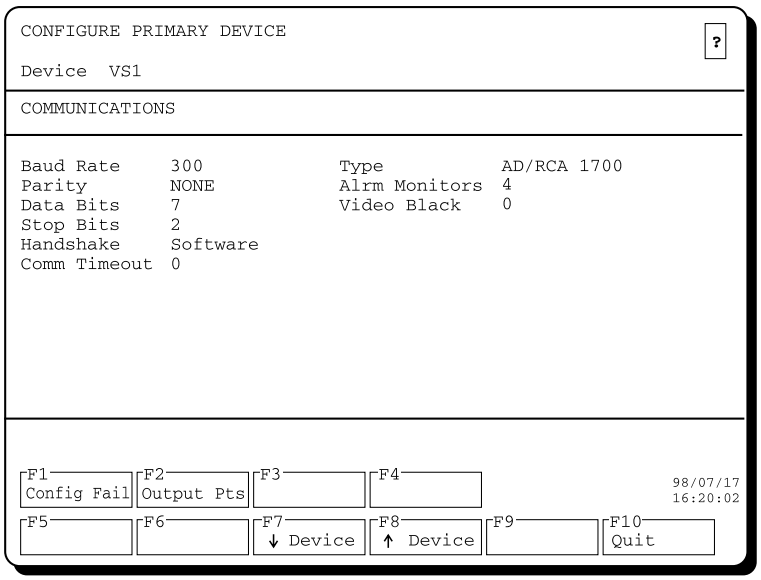
If a custom output control is being used, then enter the correct number from **1-99** in the **Custom group** field. If no custom output control is being used, then the **Custom group** field should read **0**.

Return to the **Configure points** menu by pressing **F10**. If you have not saved the output point types, you'll be prompted to do so before exiting.

For each output point, specify the point type. When you select an output point, a pop-up menu appears, listing the output point types you've created. Pick the type from the menu.

Entering data on a video switcher

The menu for the video switcher is shown below.



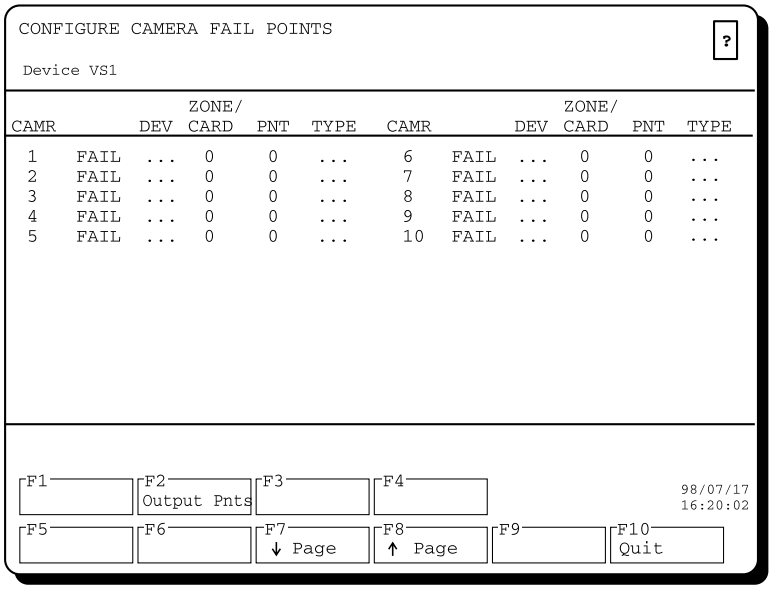
To change the default settings, select new settings from the pop-up menus, or enter a number in the input field.

In the **Monitors** field, enter a number from 1 to 8.

In the **Video Black** field, enter a number from 1 to 999. For no video black, enter 0.

Configuring points for camera fail

Select **F1 - Config Fail** from the Configure Primary Devices menu to configure points for camera fail.

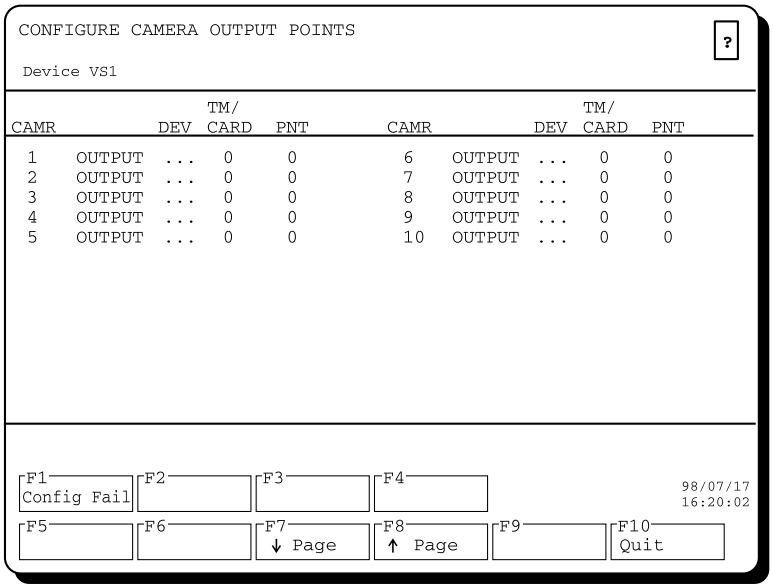


For each camera, specify the input point that generates a fail alarm. Select the **Fail** field, then enter the data in the input fields. In the Dev field, enter the 3-character device name (e.g., IU1). In the Type field, enter Detn, Tamp or Fail.

Configuring Camera output points

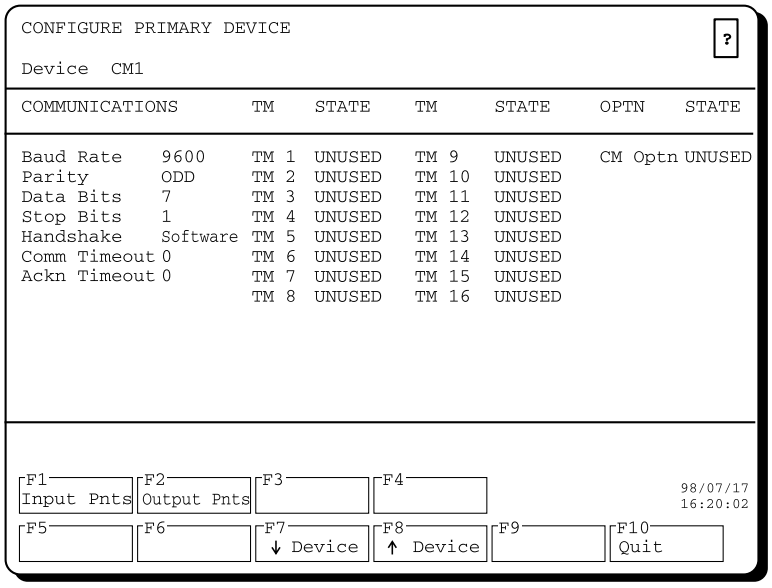
Select **F2 - Output Pnts** to configure output point for the cameras.

For each camera select the **output** point, enter the three character device name (e.g., IU1) in the Dev field, enter the TM or Card number and the point number in the two remaining fields.



Entering data on a Control Module

The menu for the Control Module is shown below.



In this menu, you:

- set the communications parameters for the Control Module
- specify what Transceiver Modules are attached to the Control Module
- specify whether you have an option card in the Control Module

The default parameters for the Control Module are shown. You can change only the baud rate and both Timeout parameters.

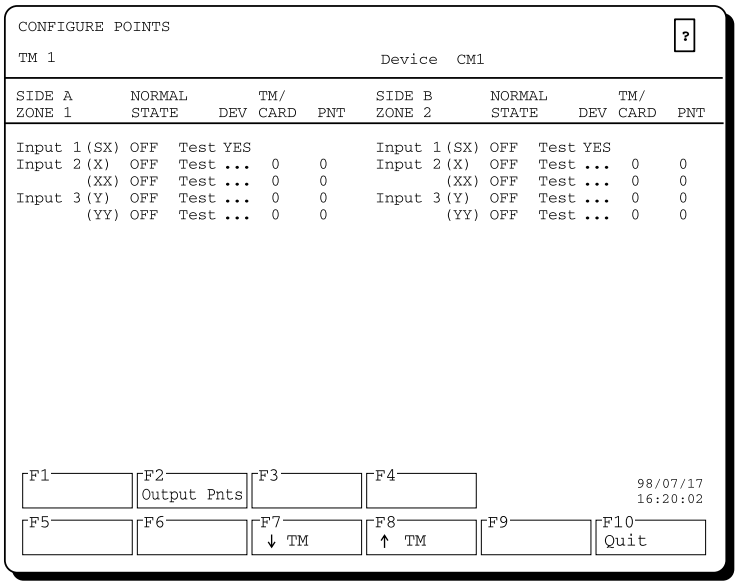
Highlight each **TM** that is connected to the Control Module and select Used from the pop-up menu.

If you have an option card installed in the Control Module, move to the **CM Optn** field and select **Used** from the pop-up menu.

Configuring input points for TMs

If you have TMs connected to the Control Module, you must configure the input points for each TM.

Select **F1 - Input Pnts**.



In this menu you specify the normal state of the inputs and the configuration of the test devices (if any).

The cursor is on **Input 1** on Side A - Sentrax/S∞Trax input. You can't change the normal state for Sentrax/S∞Trax; however, you can specify whether or not Sentrax has self-test capabilities. The default isYes.

Inputs **2** and **3** are for the auxiliary inputs at the Sentrax/S∞Trax Transceiver Module. To change the normal state for the input, select the input field and pick On from the pop-up menu.

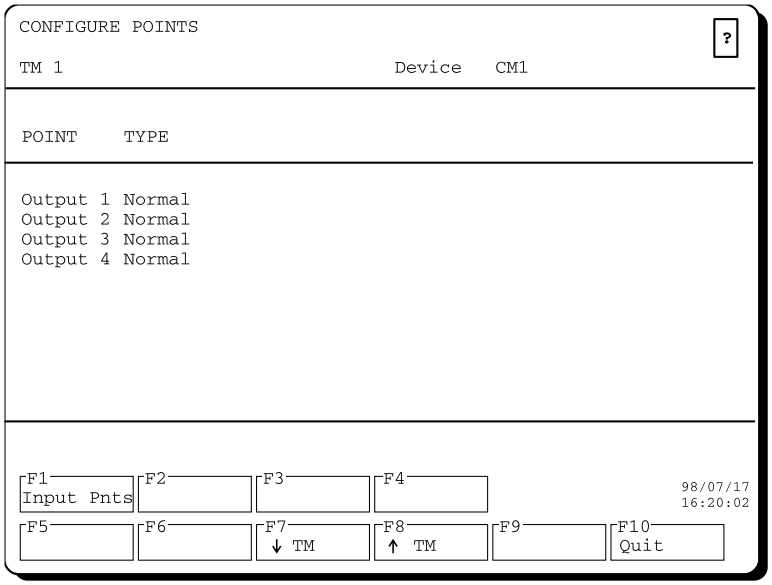
To enter a test device, select the **Test** field. Input fields appear for the test device, TM/card number and point number. In the Dev field, enter the 3-character code for the device (e.g., CM1). Then enter the TM or card number, and point number.

If you have a Data Module (DM) in the system, it is treated as a TM with the exception of Input 1 (Sentrax/S∞Trax), which is ignored. The auxiliary sensors remain as inputs 2 and 3.

Configuring output points for TMs

You must configure the output points for each TM connected to the Control Module. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

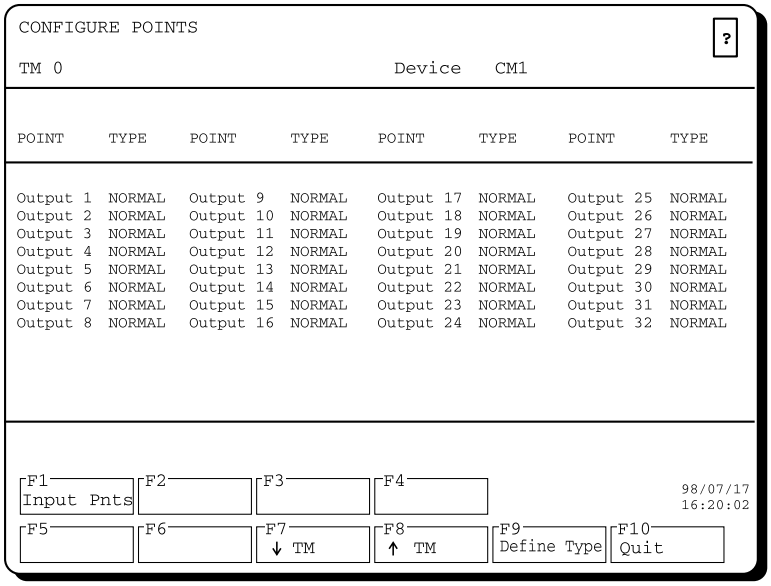
Select **F2 - Output Pnts**.



The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Configuring output points for option cards

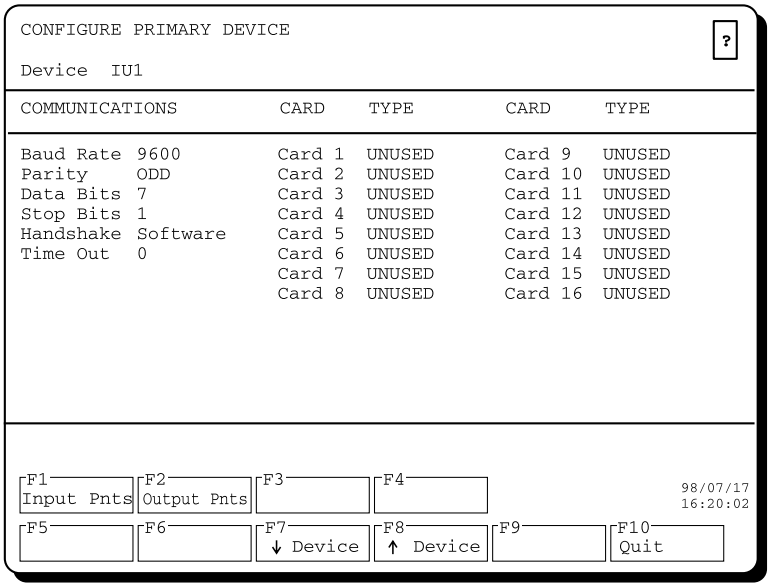
If you have a CM option card, the F7 or F8 function keys will step to the **Configure CM Option** (TM0) menu. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.



The screen displays the 32 output points with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Entering data on an Interface Unit

The menu for the Interface Unit is shown below.



In this menu you set the communications parameters for the Interface Unit, and list the collector and distributor cards.

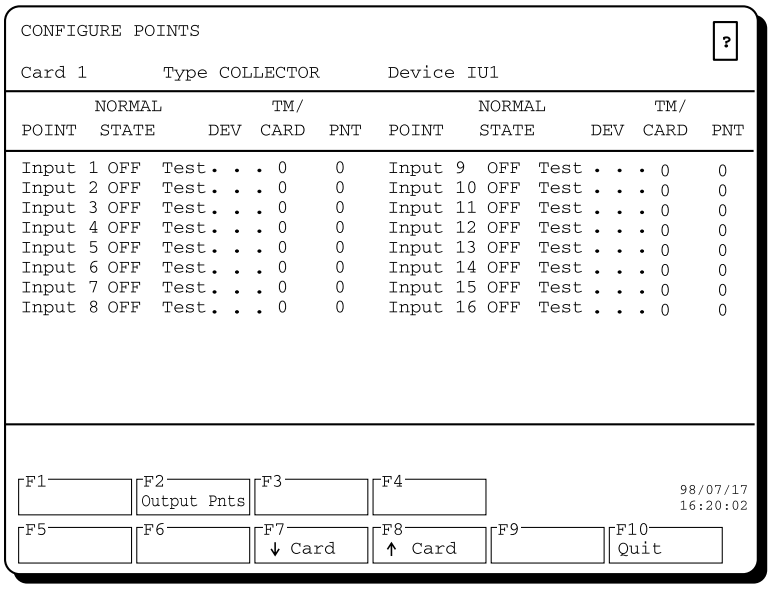
> *If you are using supervised collector cards, the alarm state appears as inputs 1 to 8, and the corresponding tamper state appears as inputs 9 to 16.*

In the **Card** column, indicate whether the cards are collector or distributor cards, by selecting each card that is installed in the Interface Unit, and specifying Collector or Distributor from the pop-up menu.

Configuring input points for a collector card

If you have collector cards installed in the Interface Unit, you must specify the normal state and test devices attached to each point on each card.

Select **F1 - Input Pnts**.



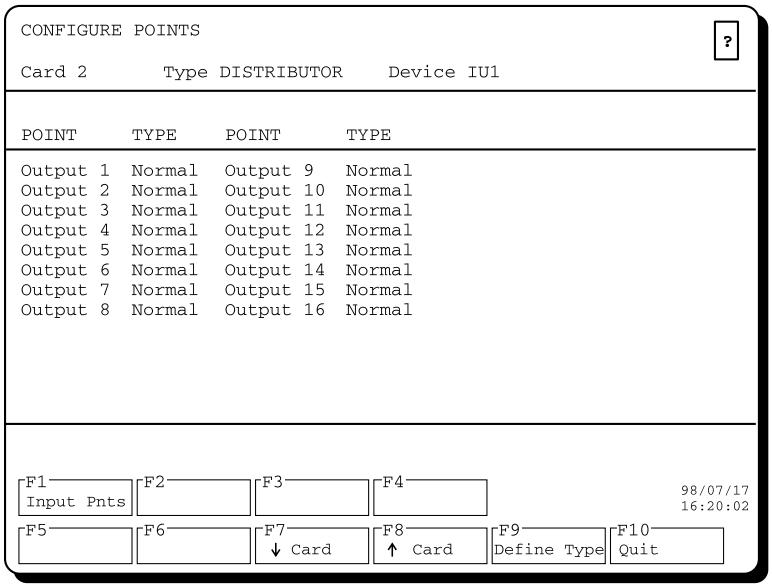
To change the default normal state for an input point, select the point and pick On from the pop-up menu.

To enter a test device for an input, select the **Test** field. Input fields appear for the test device, TM/card number and point number. In the Dev field, enter the 3-character device code (e.g., IU1). Then enter the TM or card number, and the point number.

Configuring output points for a distributor card

If you have distributor cards installed in the Interface Unit, you must configure each point on each card. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

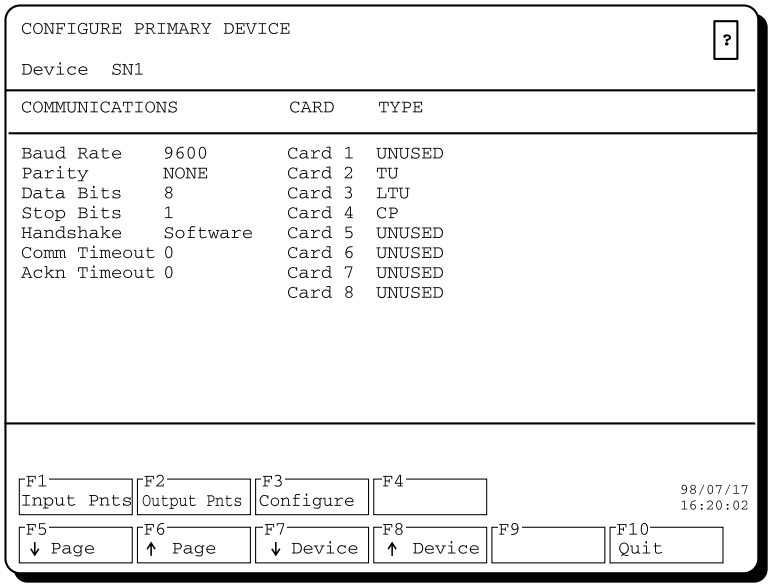
Select **F2 - Output Pnts**.



The 16 output points are listed with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Entering data on the Sennet network

The menu for the Sennet network is shown below.



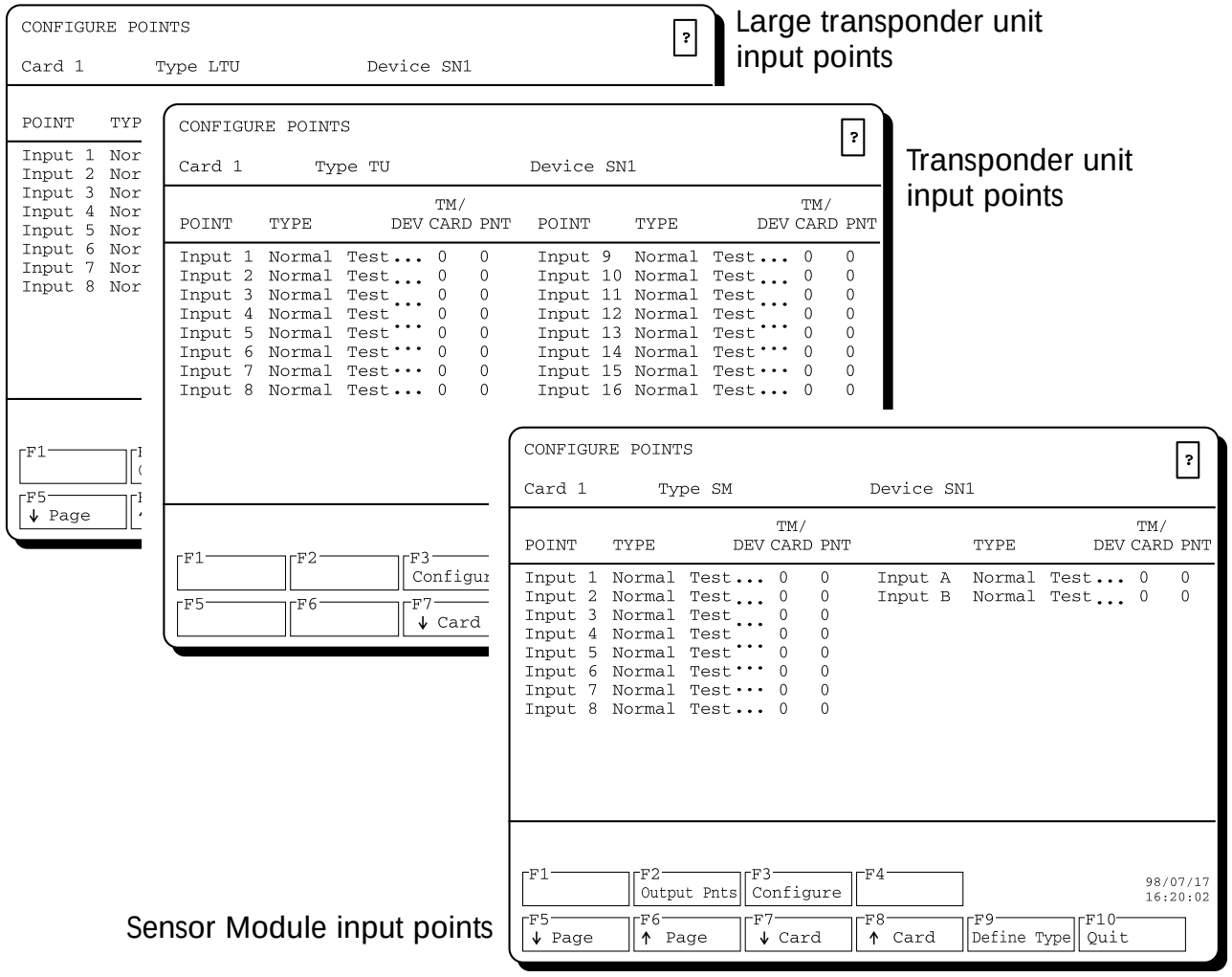
In this menu you set the communications parameters for the Sennet network and list the transponder cards.

The default parameters for the Sennet network are shown. You can change only the baud rate and timeout settings.

In the **Card** column, select each card that is installed in the Sennet network and specify TU, LTU, SM or CP from the pop-up menu.

Configuring input points for the Sennet network

To configure input points for the LTU, TU or SM, press **F1 - Input Pnts**.



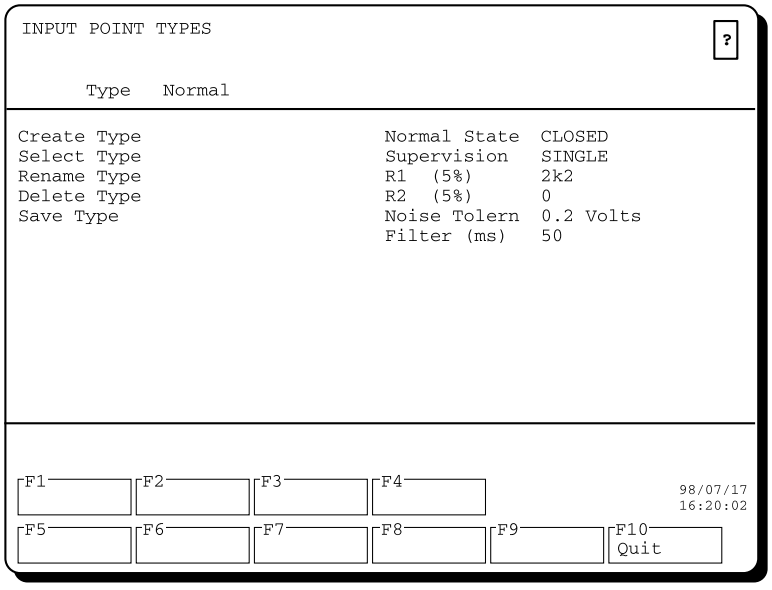
In this menu, you:

- define the input point types
- specify input point test points

Input points are defined in terms of the following parameters: normally open/closed contacts, none/single/dual resistor supervision, noise tolerance level, and debounce filter time. An input point **type** is defined as a combination of these parameters. When you define an input point type, you can then assign a type to each of the transponder input points. This saves you from having to enter each parameter individually for each input point.

Defining input point types

To define the input point types, press **F9 - Define Type** from the input points menu.



The top line displays the input point type that you're defining. In this case, a system default type **Normal** has already been created.

To create a new input point type, enter a type name in the **Create Type** field. Names can be up to eight characters, including spaces. You can create up to eight input point types for the system.

For each input point type, specify:

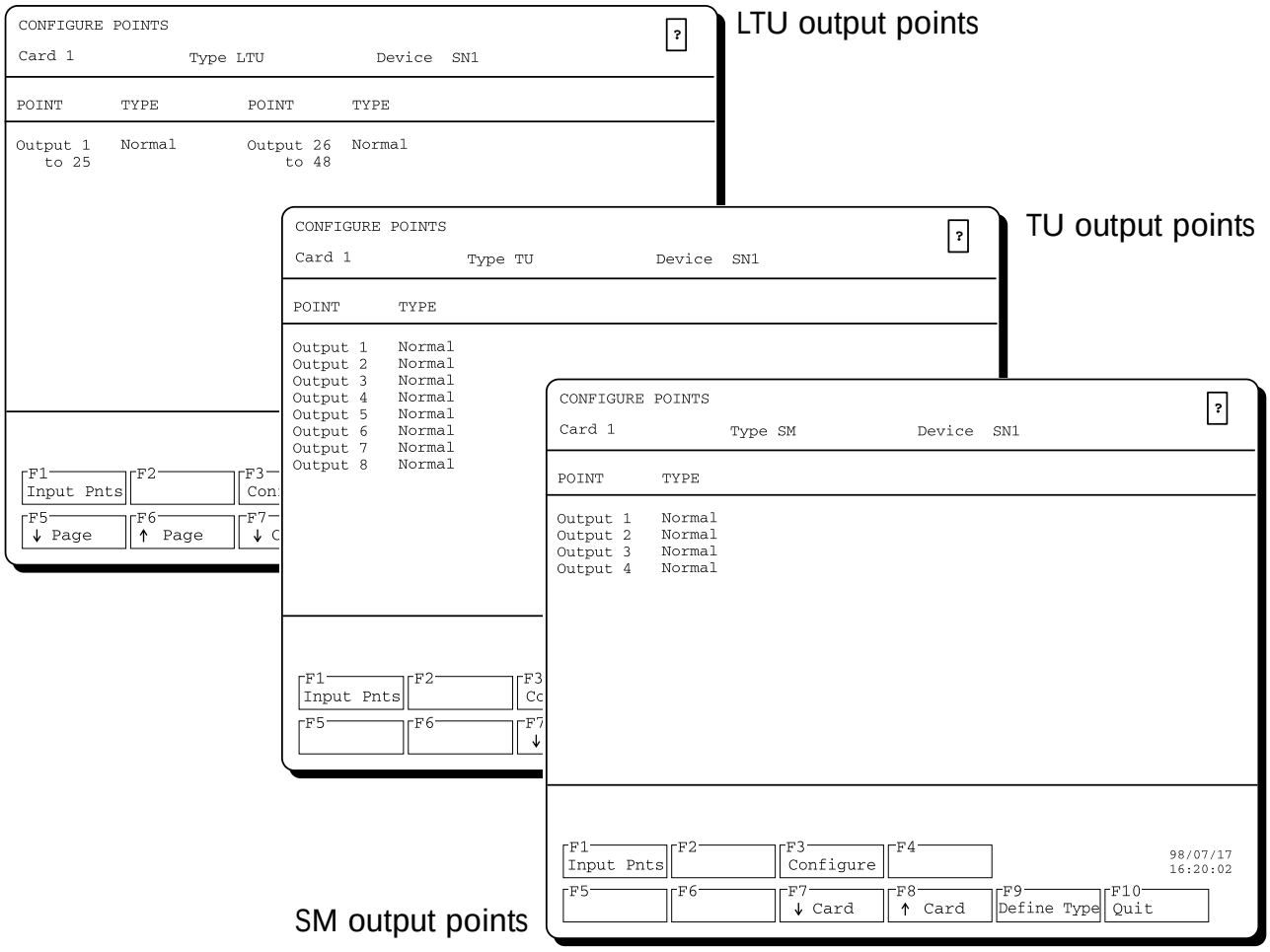
- normal state — specify **Closed** or **Open**
- supervision — if you select **None**, the R1, R2 and Noise Tolern fields are automatically set to 0, and cannot be changed. If you select **Single**, you can change the R1 resistor value and the noise tolerance. If you select **Dual**, the R1 and R2 fields are automatically set to 2k2 and cannot be changed, but you can change the noise tolerance.
- filter debounce time — enter a value from 0 to 2500 milliseconds (in increments of 20)

Return to the Configure input points menu by pressing **F10**. If you have not saved the input point types, you'll be prompted to do so before exiting.

For each input point, specify the input point type and test device (if applicable). When you select the input point, a pop-up menu appears, listing the input point types you've created. Pick the type from the menu.

Configuring output points for the Sennet network

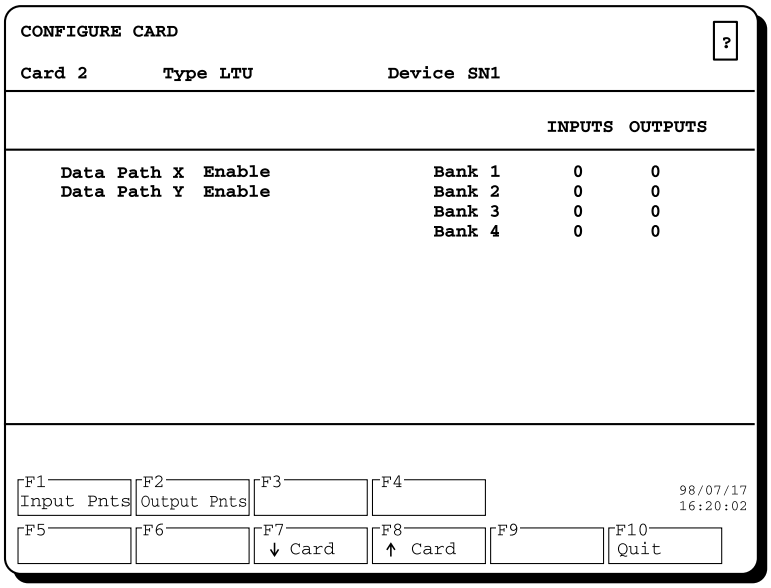
To configure output points for the LTU, TU or SM, press **F2 - Output Pnts**. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.



Output point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Configuring cards for a large transponder unit

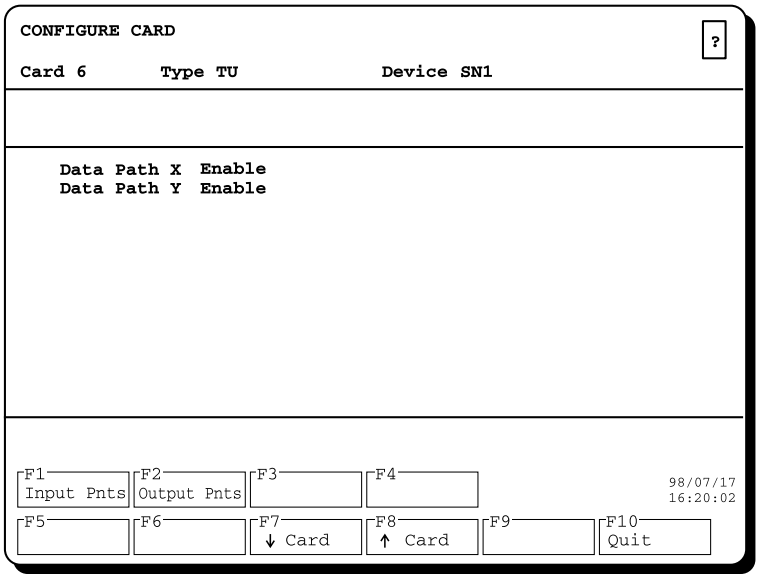
To configure the large transponder unit card, press **F3 - Configure**. Using the scroll functions **F7** and **F8** select an LTU card.



From the pop-up menus select the inputs and output for each bank as required.

Select **Data Path X** and Enable or Disable as required. Repeat this for Data Path Y.

Configuring cards for a transponder unit

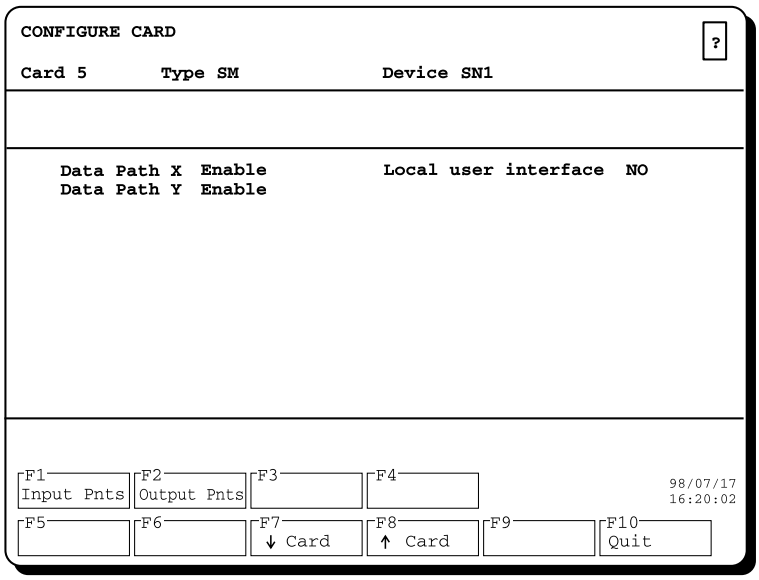


To configure the transponder unit card, press **F3 - Configure**. Using the scroll functions **F7** and **F8** select a TU card.

Select **Data Path X** and Enable or Disable as required. Repeat this for Data Path Y.

Configuring cards for a Sensor Module

To configure the Sensor Module card, press **F3 - Configure**. Using the scroll functions **F7** and **F8** select an SM card.

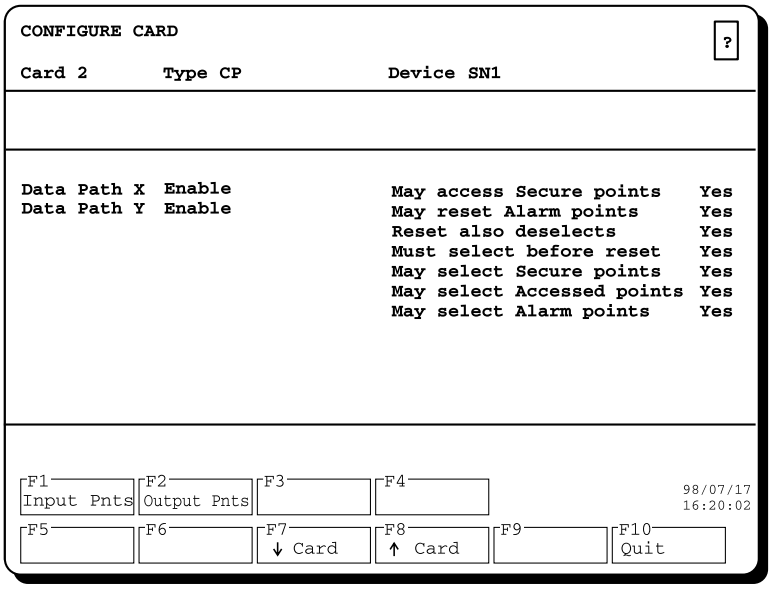


Select **Data Path X** and Enable or Disable as required. Repeat this for Data Path Y.

Select the **Local User Interface** and set On or OFF as required.

Configuring cards for a control panel

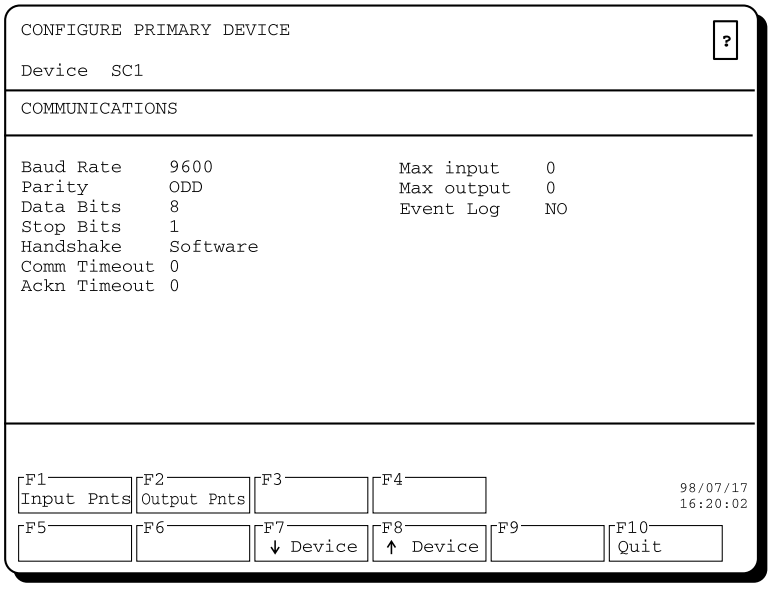
Press **F3 - Configure**. Using the scroll functions **F7** and **F8** select a CP card.



Select **Yes** or **No** for each option.

Entering data on Starcom devices

The menu for Starcom devices is shown below.



The default parameters are shown. You can't change the Data Bits field.

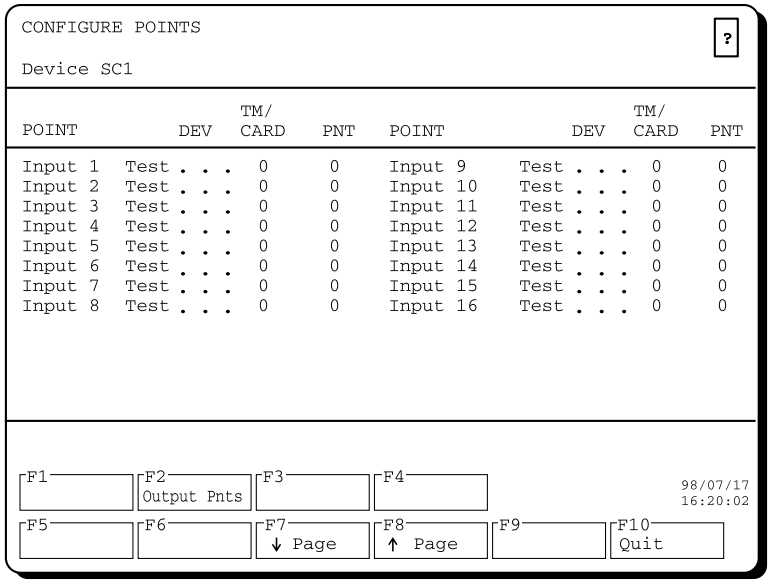
To change the other default settings, pick another setting from the pop-up menus, or enter a number in the input field.

The maximum number of input and output points can be set from 0 to 1024 each.

Configuring input points

For each input point connected to a Starcom device, you must specify whether there is a test device attached.

Select **F1 - Input Pnts**.

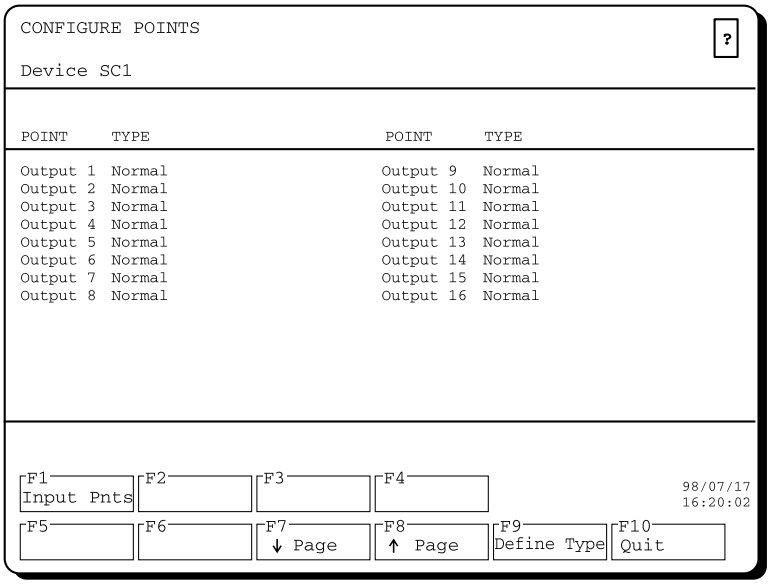


Select an input point and fill in the device name, TM or card number, and point number in the input fields.

Configuring output points

You must configure each output point connected to a Starcom device. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

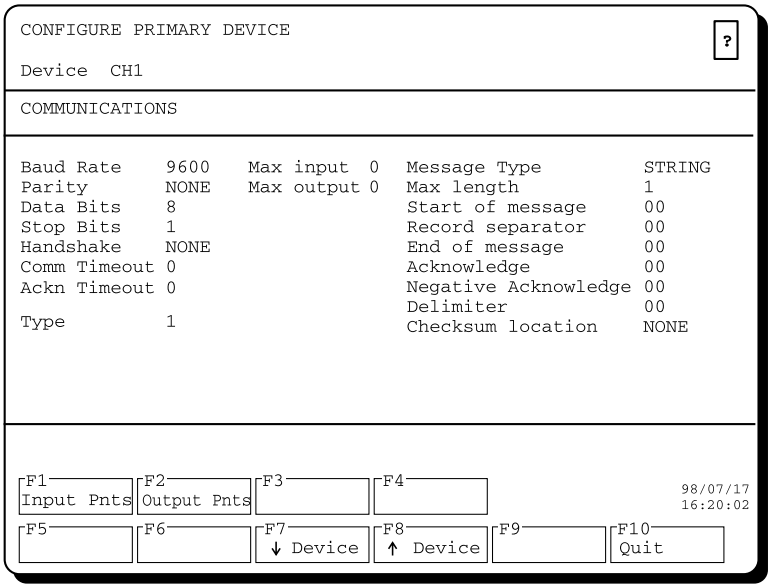
Select **F2 - Output Pnts**.



The output points are listed with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Entering data on custom devices

The menu for custom devices is shown below.

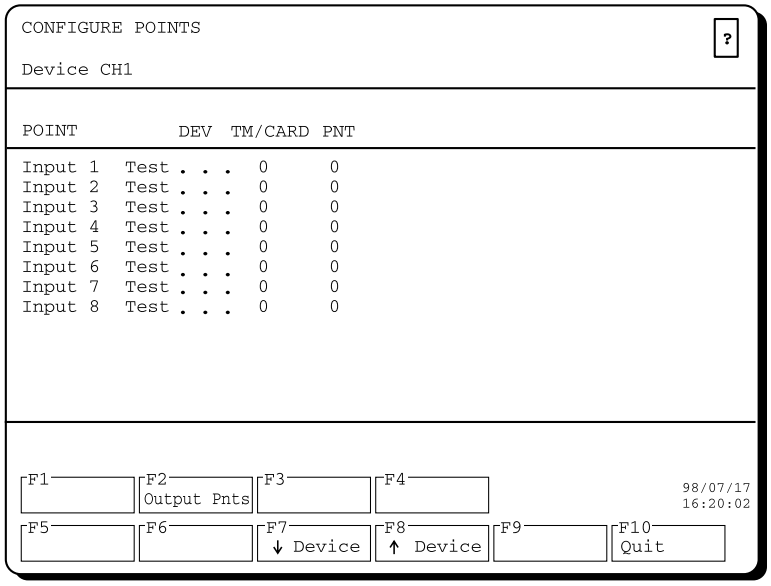


The default parameters are shown. To change the default settings, pick another setting from the pop-up menus, or enter a number in the input field.

Configuring input points

For each input point connected to a custom device, you must specify whether there is a test device attached.

Select **F1 - Input Pnts**.

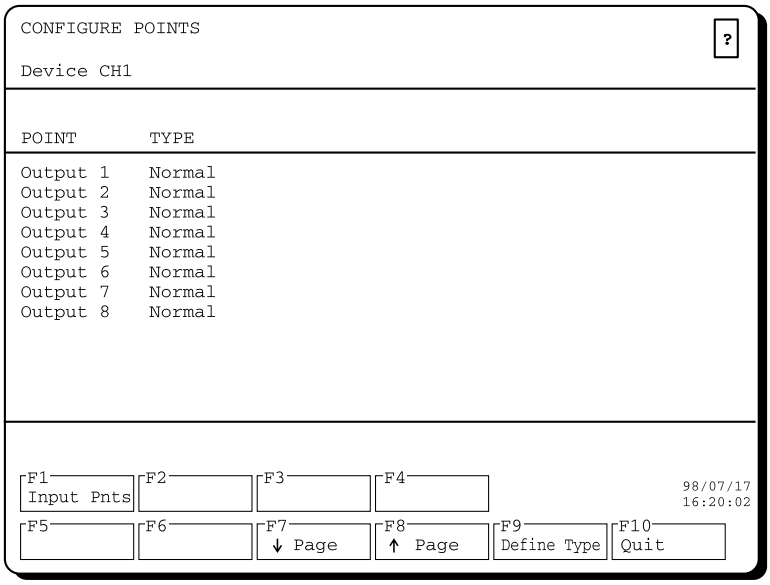


Select an input point and fill in the device name, TM or card number, and point number in the input fields.

Configuring output points

You must configure each output point connected to a custom device. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

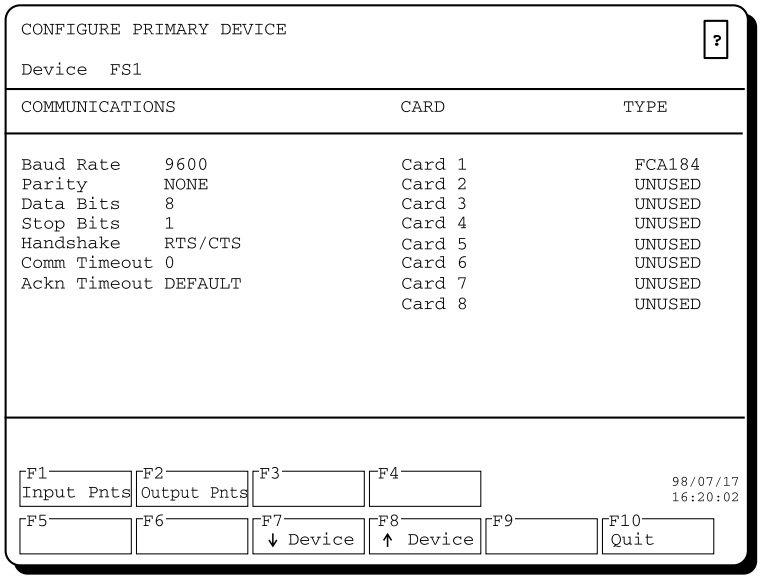
Select **F2 - Output Pnts**.



The output points are listed with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Entering data on Fiber sensys devices

The menu for Fiber sensys devices is shown below.

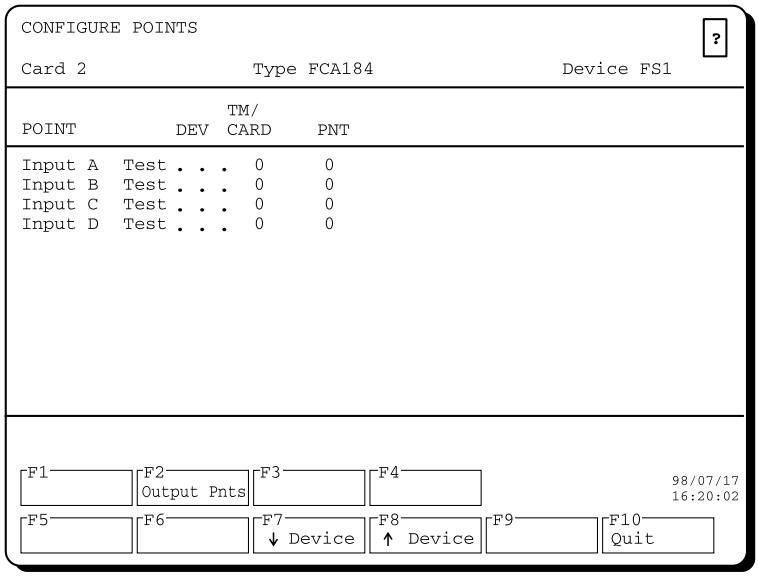


The default parameters are shown. To change the default settings, pick another setting from the pop-up menus, or enter a number in the input field.

Configuring input points

For each input point connected to a Fiber sensys device, you must specify whether there is a test device attached.

Select **F1 - Input Pnts**.

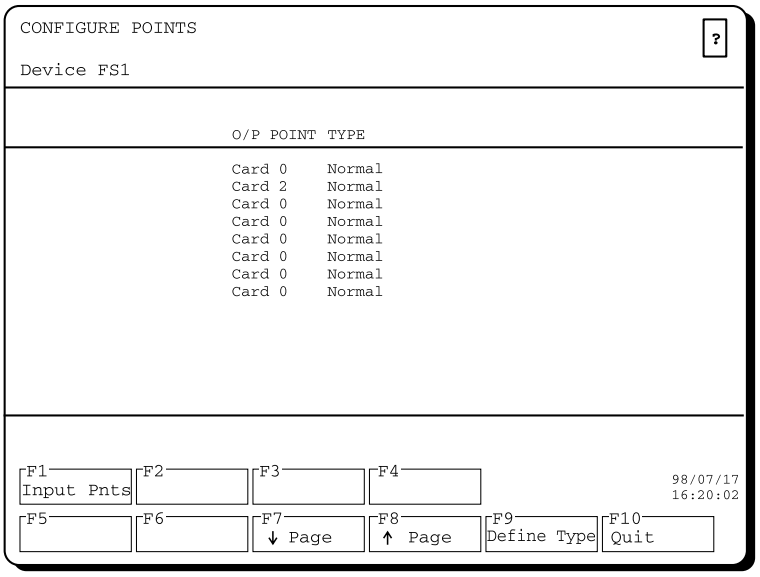


Select an input point and fill in the device name, TM or card number, and point number in the input fields.

Configuring output points

You must configure each output point connected to a Fiber sensys device. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

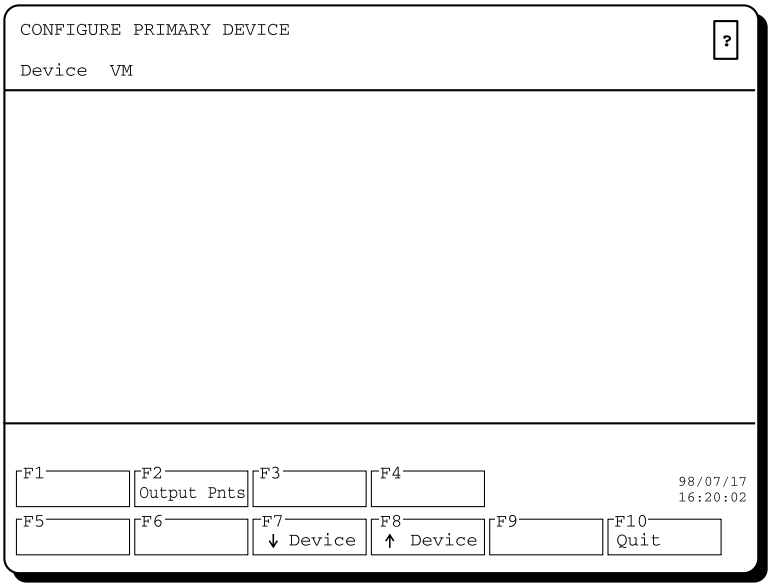
Select **F2 - Output Pnts**.



The output points are listed with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Entering data on the virtual memory device

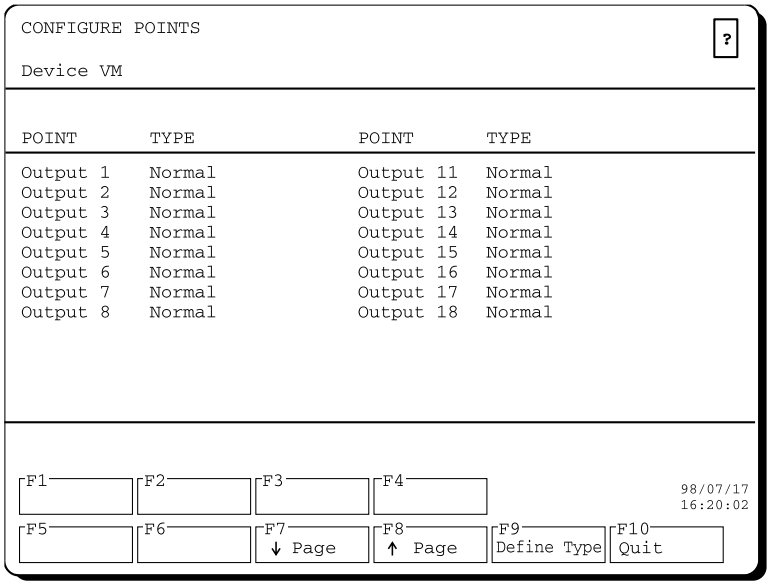
The menu for the virtual memory device is shown below.



Configuring output points

You must configure each output point connected to the virtual memory device. Configure the output points as described in ***Configuring output points for a Control Unit***, page 3-12.

Select **F2 — Output Points**.

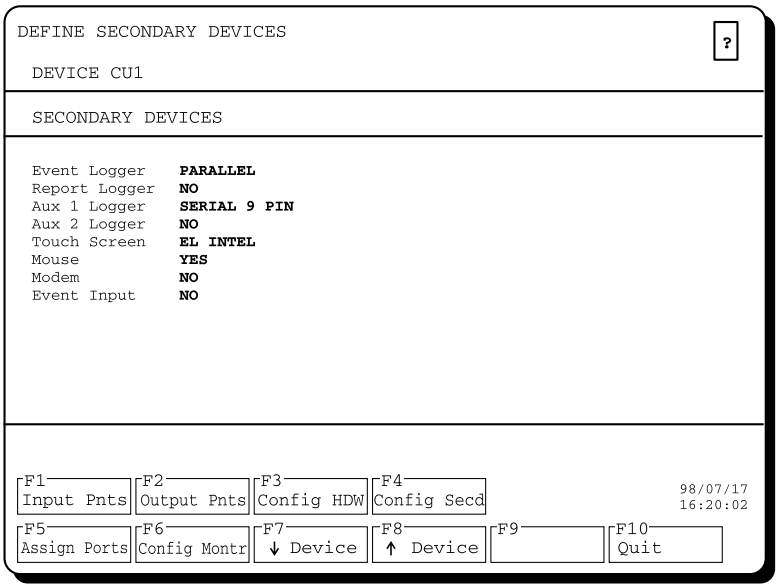


The output points are listed with the default type. The point types are defined in the same way as described in ***Defining output point types***, page 3-13.

Configuring secondary devices

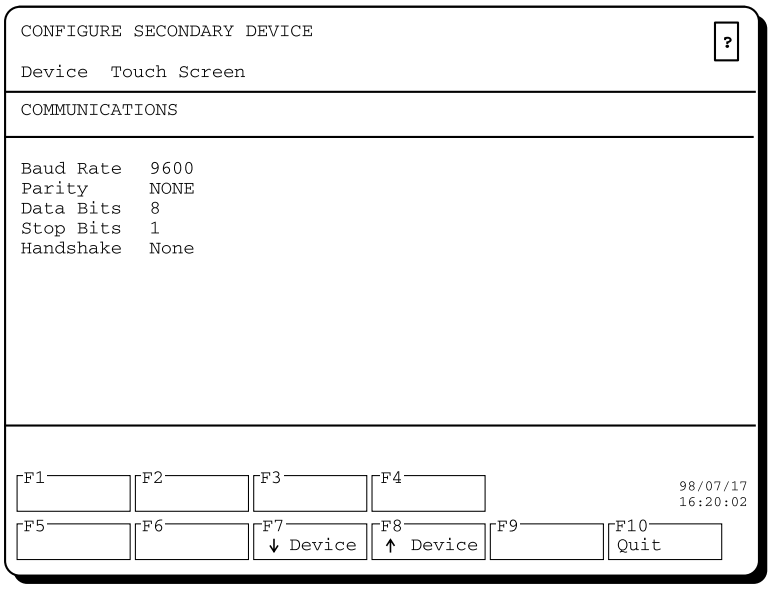
For each secondary device in your system, you must specify the communications parameters (baud rate, parity, data bits and stop bits).

To enter the data, press **F4 - Config Secd**, from the Define Secondary Devices menu.



The menu for the first serial device in the list is displayed. If your event logger is a serial printer, the first menu displayed will be for the event logger. If your event logger is a parallel printer, you don't specify any communications parameters; therefore, the menu displayed will be for the next device in the list.

A sample menu is shown below.



The default parameters for the selected touch screen are shown. To change any of the parameters, select the parameter and pick another value from the pop-up menu.

A similar menu is available for all of the secondary devices.

Saving hardware configuration data

Once you've entered all the hardware data in the database, and reviewed the port assignments, ensure that you save all the data.

To save the configuration data:

1. Return to the Configure Hardware menu and select the **Save Configuration** field. A pop-up menu appears, asking you to confirm that you want to save the data.
2. Select **Yes** to save. The system displays a message on the prompt line when the save process is complete.

> *Do not cancel the function once it has started to save.*

If you attempt to exit from the Configure Hardware menu without saving your work, a pop-up menu appears, reminding you that you haven't saved. You have the option of saving and quitting, quitting without saving, or cancelling the function. This feature provides added security, so you don't forget to save your input.

Editing hardware configuration data

At any time you can enter the Configure Hardware menus and change the configuration data. You might want to do this if:

- you've added a new card or device to your system
- you've forgotten to enter a card or device in the database
- you've modified the configuration of any of the devices

To edit hardware data, re-enter the menu for the affected device and input the new data. Make sure that you enter information in all the applicable menus. For example, if you add a CM option card to your system, you must specify it in the CM menu and configure the option card points in the option card menu. Remember to save your changes before exiting the Configure Hardware menu, otherwise the data reverts to the way it was at the last save.

If you make any changes that will affect the port assignments (i.e., add a new device or card), you must review the port assignments, so that the system can reassign devices and ports if necessary.

4 Drawing maps

Overview

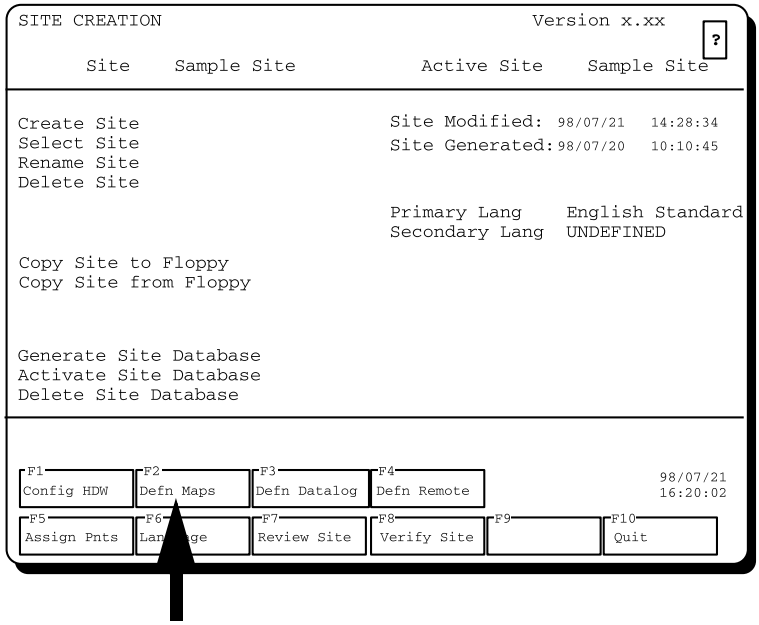
As part of your site plan package, you should have included hand-drawn maps of your site. These maps depict the area you want to monitor. For example, a map could be a floor plan of a building, or the outdoor area surrounding a building, or a combination of indoor and outdoor areas. Depending on your system size, you can have up to 64 maps for your site.

The hand-drawn maps now need to be drawn in the Site Creation package. This chapter explains how to:

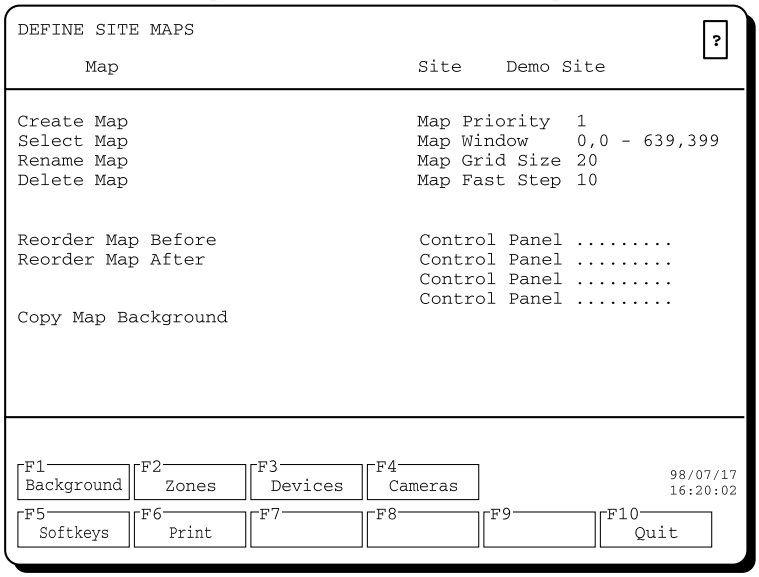
- draw the maps on the color screen
- put the zones and sensors on the maps
- put the device symbols on the maps
- indicate the camera locations on the maps

Entering the Defn Maps menu

To draw maps on the primary screen, select **F2 - Defn Maps** from the Site Creation main menu.



The screen displays the Define Site Maps menu.



The **Site** field holds the name of the site database that you created in the main menu. The **Map** field is blank until you create a map, or select one from a list of created maps.

Creating a new map

To create a new map, you must first name it, then set certain specifications for it, which you'll use when drawing the map. These specifications have default settings, and you may want to keep the defaults.

The **Create Map** function is highlighted when you enter the menu. To create a new map, press the **pick** key, then enter the map name in the input field. Map names can be up to 15 characters, including spaces. If you key in an existing map name, you'll get an error message.

The new map name appears in the **Map** field, in the current-selection area of the screen.

Setting the map scale

You draw maps on the color-graphics monitor. The map window is fixed to the size of the color screen. The default coordinates

(x coordinate: 0-639 units; y coordinate: 0-399 units) appear next to the Map Window function. You can change the scale to any coordinate between ±999. Generally, the larger the scale, the more detailed you can make the maps. However, the amount of detail visible depends on the resolution of the display card. (EGA 640 x 350, VGA 640 x 480)

To set the map scale:

1. Select the **Map Window** function. Input fields open for the x and y coordinate. The first two fields are the start points for x and y, and the last two fields are the end points.
2. Enter a number in the field and press the **Tab** key to move on to the next field. Press **Enter** when done.

The map on the primary screen is rescaled to reflect the new window size.

Setting the map grid increments

When you draw a map, you have the option of using a grid to assist in accurate sizing and positioning of objects. This grid is placed over the map and can be toggled on and off. You can set the grid increments to any number. For example, a setting of 20 means that a grid line will appear every 20 units. A setting of 5 means that a grid line will appear every 5 units; therefore you will have more grid lines on the screen. The more grid lines you have on the screen, the easier it is to size or position the objects on the map.

To set the grid increments, select the **Map Grid Size** function and enter a number in the input field.

Setting the cursor speed

You have the option of changing the cursor speed on the color monitor. The Map Fast Step option lets you set the speed of the cursor for fast movement. The default setting is 10, which means that every time you press the cursor key in fast step mode, the cursor steps over 10 units. The higher the setting, the more area the cursor will cover in one move (i.e., the faster the cursor will move).

To set the cursor speed, select the **Map Fast Step** function and enter a number in the input field.

Selecting a map priority

The Map Priority function sets the priority of the current map in relation to the other maps for the site.

Map priority is an option that you may need if you have certain areas where security is more important. For example, if your maps are floor plans of your building, the map of the computer section might be considered a higher priority than the map of the storerooms. The map

priority is added to the individual sensor priority. When two or more alarms occur at the same time, the higher priority alarm can be viewed first.

The default priority is 1, or lowest priority. To assign another priority to the current map, select the **Map Priority** function and enter a number between 1 and 64 (64 is the highest priority).

Reordering maps

When you pick the Select Map function, a pop-up menu appears, listing the maps in the order that you've created them. When the operator scrolls through the maps in the Operator menu, they'll also see the maps in this order.

You can change the order that the maps will appear in by selecting the Reorder Map Before or Reorder Map After function.

To change the position of a map in the list:

1. Choose the **Select Map** function and pick a map from the pop-up menu. The map name appears in the Map field and this map becomes the current map.
2. Select the **Reorder Map Before** or **Reorder Map After** function. A pop-up menu appears, listing the maps in their current order.
3. Pick a map from this list. If you selected the Reorder Map Before function, the current map (displayed in the Map field) will be positioned before the map you've picked. If you selected the Reorder Map After function, the current map will be placed after the map you've picked.

Editing maps

You can modify any of the specifications for the map - map window, grid size or fast step, by selecting the function and inputting the new data.

You can rename or delete the current map by selecting the corresponding function. You can also select another map to edit.

Assigning maps to Sennet Control Panels

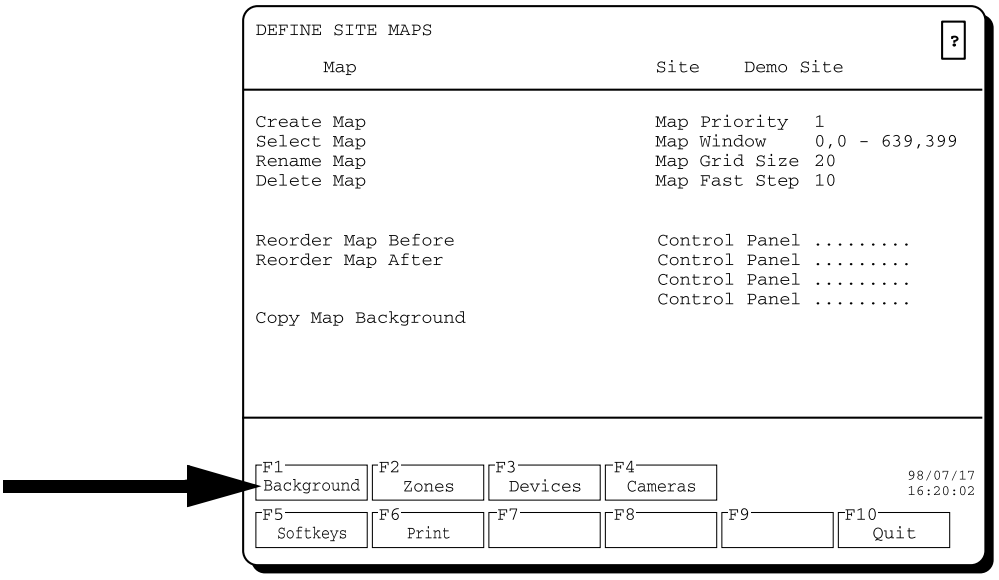
You can assign a map to a maximum of 4 Control Panels, and a maximum of 4 maps to any one Control Panel as long as the Control Panel has no more than 255 points assigned to it.

When a map has been assigned to a Control Panel the alarms displayed on that map can be controlled from and annunciated at the Control Panel.

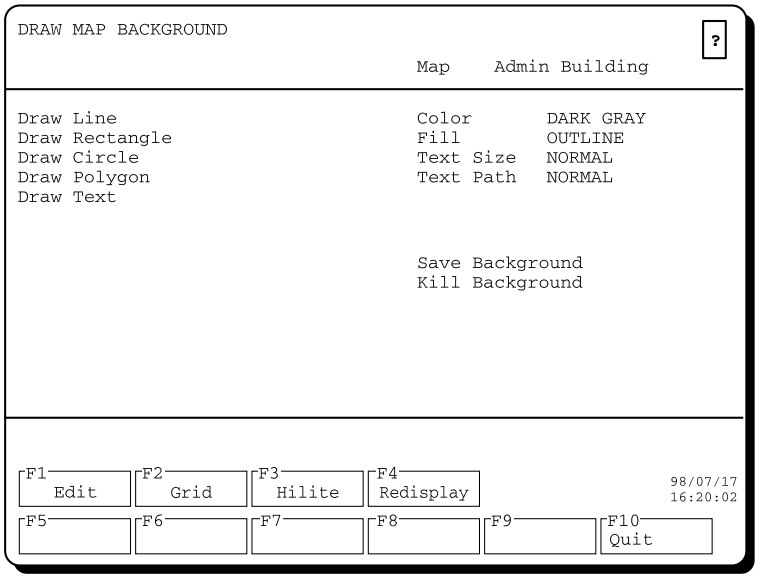
Entering the Map Background menu

Once you've created a new map and set the specifications for it, you are ready to draw it on the primary monitor. The first step is to draw all identifying objects such as buildings, room layouts, trees, etc. These objects are referred to as the map background.

Select **F1 - Background** from the Define Site Maps menu.



The screen displays the Map Background menu.



The **Map** field holds the name of the current map, selected in the previous menu.

In this menu you draw your background objects by selecting the shape that you want to draw and then drawing it on the color screen. Refer to your hand-drawn map to ensure that the objects are positioned properly and are to scale.

Changing the appearance of objects

To draw your maps you can use lines, rectangles, circles, polygons and text. For each shape that you draw, you can set certain options such as color, fill, text size, and direction of text. The options appear in the column on the right hand side of your monochrome screen. To change any of these options once you've drawn the object, use the Edit function. See ***Editing the map background***, page 4-18.

Drawing map objects

This section explains how to use all the available shapes for drawing - lines, rectangles, circles, polygons and text. As you draw the shapes on the color screen, you'll be prompted at each step. The prompts appear in the function area of the secondary screen.

Using the function keys

When you select an object to draw, the following functions appear. These functions affect the cursor on the primary screen.

Function	Description
F1 - Slow/Fast	lets you control the speed of the cursor on the primary screen. F1 is a toggle between fast (which moves the cursor the number of units you specified in Map Fast Step), and slow (which moves the cursor one coordinate at a time). The cursor speed is displayed at the bottom left of the primary monitor.
F2 - Grid	places a grid over the map on the primary screen. Pressing F2 again removes the grid.
F7 – F8	moves the cursor to the next and previous points on the object you're drawing (affects only lines and polygons)
F10 - Quit	returns control to the secondary screen

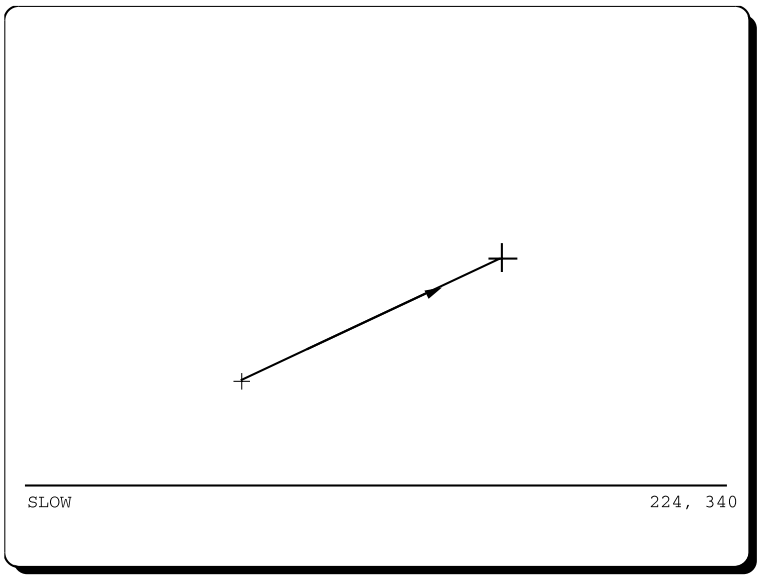
Use the functions in the table below to modify the objects on your map. These functions appear when control is at the secondary screen.

Function	Description
F1 - Edit	allows you to edit the objects on the primary screen (See ***Editing the map background***, page 4-18.)
F2 - Grid	places a grid over the map on the color screen. Pressing F2 again removes the grid.
F3 - Hilite	highlights the last object drawn on the color screen. Pressing F3 again turns the highlight off.
F4 - Redisplay	redraws the map on the color screen. Use this function if the map becomes distorted as you add more objects to it, or toggle the grid on and off.
F10 - Quit	returns to the Define Site Maps menu

Drawing a line

To draw a line on the primary screen:

1. Select the **Draw Line** function. The cursor appears on the primary screen, and you're prompted to enter a start point.
2. Use the mouse or the cursor keys to position the cursor on the primary screen, then press the **pick** key.
3. Use the mouse or the cursor keys to position the cursor where you want the line to end. A line is drawn on the screen as you move.
4. Press the **pick** key to select the end point for the line.
5. You're prompted to enter another end point. You can extend the line, move in another direction, etc. Press the **pick** key at the next end point.
6. When you've completed the line (i.e., picked the last end point), press the **done** key.



7. You're prompted to enter a start point again. This allows you to draw another line on the screen, without having to return to the secondary monitor. If you want to return to the secondary screen, press the **done** key at this prompt, or press the **F10** function key.

Erasing the line

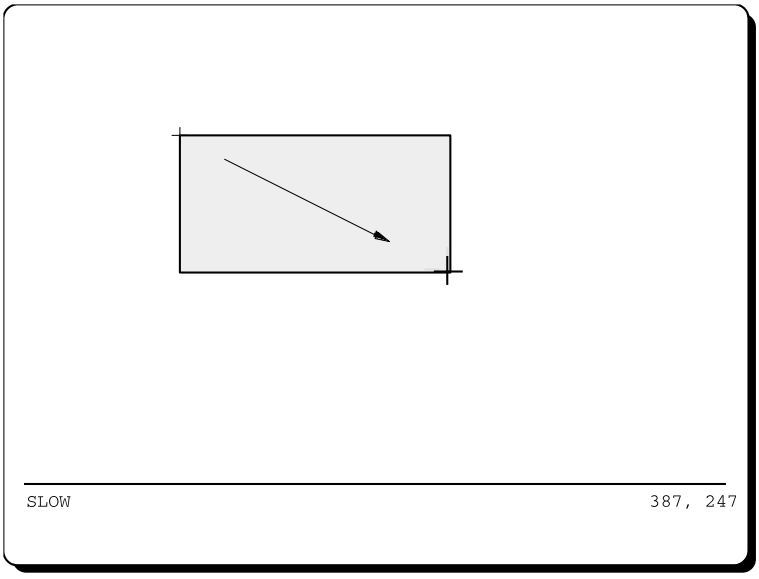
If you want to erase the line, you must erase it before you press the done key, and while the cursor is still positioned on it.To erase the line, press the **undo** key. The undo key erases the last point entered. The cursor is then positioned on the previous point. Keep pressing the undo key until all the points are erased.

To complete the function at any time, press **F10**. You'll be returned to the monochrome screen.

Drawing a rectangle

To draw a rectangle on the primary screen:

1. Select the **Draw Rectangle** function. The cursor appears on the primary screen, and you're prompted to enter a start point.
2. Use the mouse or the cursor keys to position the cursor on the primary screen, then press the **pick** key.
3. Select a diagonal point on the screen. A rectangle appears on the screen as you move the cursor.
4. Press the **pick** key when the rectangle is the size you want.



5. You're prompted to enter a start point again. You can draw another rectangle or return control to the secondary screen by pressing the **done** key or **F10**.

Erasing the rectangle

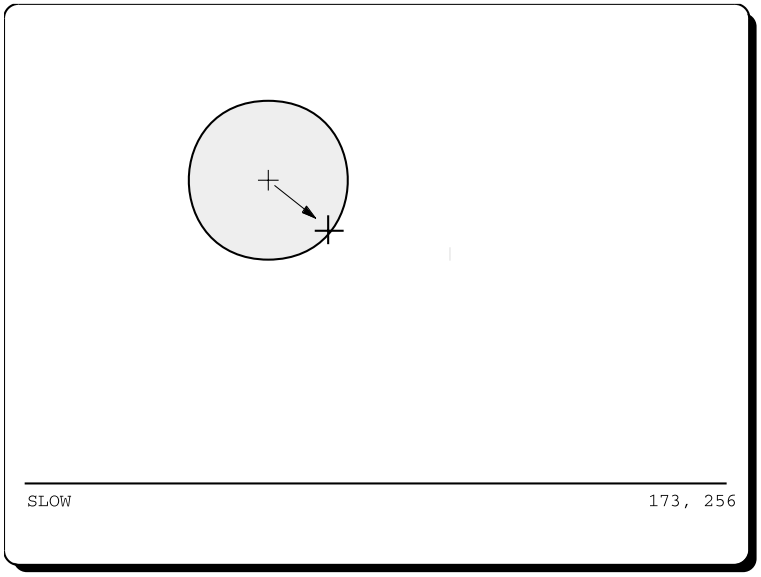
If you want to erase the rectangle, you must erase it before you pick the diagonal point, and while the cursor is still positioned on it.To erase the rectangle, press the **undo** key.

To complete the function at any time, press **F10**. You'll be returned to the secondary screen.

Drawing a circle

To draw a circle on the primary screen:

1. Select the **Draw Circle** function. The cursor appears on the primary screen, and you're prompted to pick the center point of the circle.
2. Use the mouse or cursor keys to position the cursor on the primary screen, then press the **pick** key.
3. Select the radius of the circle. A circle appears on the screen as you move the cursor.
4. Press the **pick** key when the circle is the size you want.



5. You're prompted to enter a start point again. You can draw another circle or return control to the secondary screen by pressing the **done** key or **F10**.

Erasing the circle

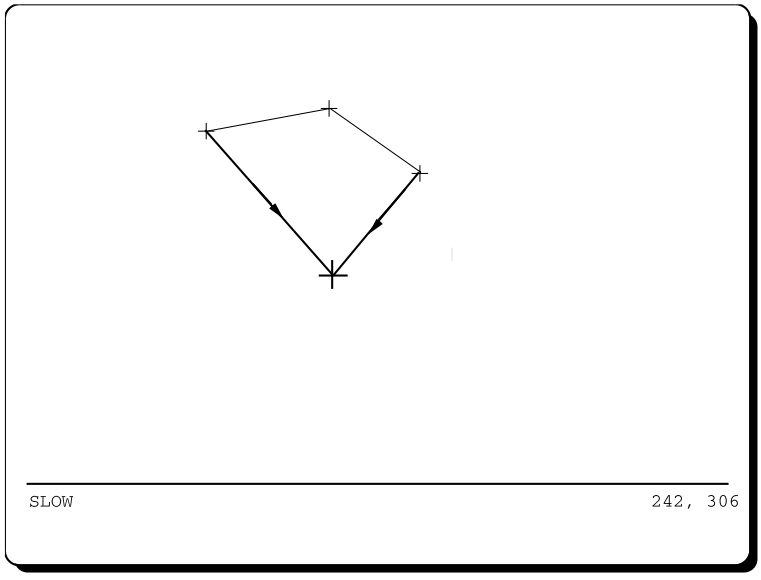
If you want to erase the circle, you must erase it before you pick the radius point, and while the cursor is still positioned on it.To erase the circle, press the **undo** key.

To complete the function at any time, press **F10**. You'll be returned to the secondary screen.

Drawing a polygon

To draw a polygon on the primary screen:

1. Select the **Draw Polygon** function. The cursor appears on the primary screen, and you're prompted to pick the start point.
2. Use the mouse or cursor keys to position the cursor on the primary screen, then press the **pick** key.
3. Use the mouse or cursor keys to select the second point of the polygon. A line is drawn on the screen as you move.
4. Press the **pick** key to select the end point of the line.
5. Select another point of the polygon. As you move, lines are drawn on the screen to connect the last and first points to the cursor. Continue this procedure until the polygon is the shape you want.



6. When you've completed the polygon (i.e., picked the last point), press the **done** key.
7. You're prompted to enter a start point again. You can draw another polygon, or return control to the secondary monitor by pressing the **done** key or **F10**.

Erasing the polygon

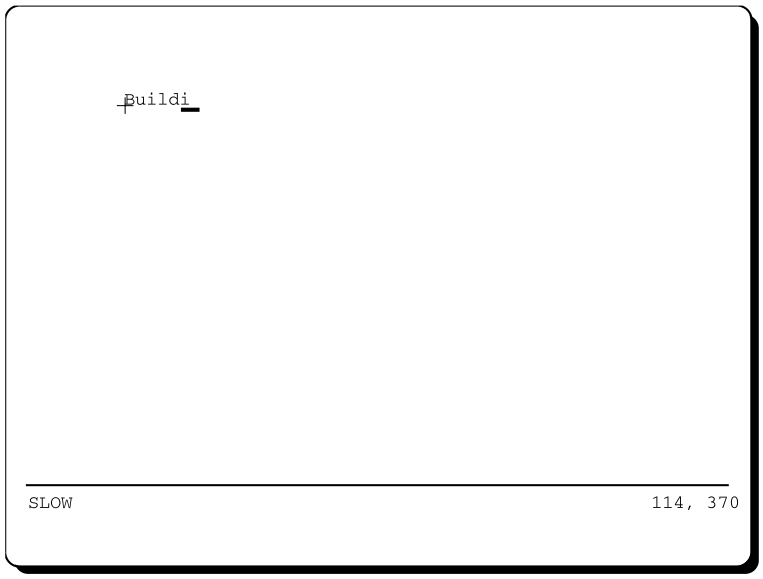
If you want to erase the polygon, you must erase it before you press the done key, and while the cursor is still positioned on it.To erase the polygon, press the **undo** key. The undo key erases the last point entered. The cursor is then positioned on the previous point. Keep pressing the undo key until all the points are erased.

To complete the function at any time, press **F10**. You'll be returned to the secondary screen.

Entering text

To enter text on the primary screen:

1. Select the **Draw Text** function. The cursor appears on the primary screen, and you're prompted to enter a start point.
2. Position the cursor on the primary screen, then press the **pick** key. The cursor changes to an underline.
3. Enter the text string. Use the **Del** and ← keys to delete text, and the ← and → keys to move backwards and forwards in the text string.
4. Press the **Enter** key after keying in the text.



5. You're prompted to enter a start point again. You can enter more text, or return control to the secondary screen by pressing the **done** key or **F10**.

To complete the function at any time, press **F10**. You'll be returned to the secondary screen.

Saving the map background

Once you've drawn your map background, ensure that you save it. If you try to exit from the Map Background menu without saving your work, you'll be prompted to save before exiting.

To save a map background, select the **Save Background** function and pick **Yes** to save. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

Deleting the map background

To delete the whole map background, select the **Kill Background** function and pick **Yes** from the pop-up menu.

CAUTION	*Once you delete a map background, you cannot recover it. This function deletes the whole background even if you've previously saved it with the Save Background command.*

Copying a map background to another map

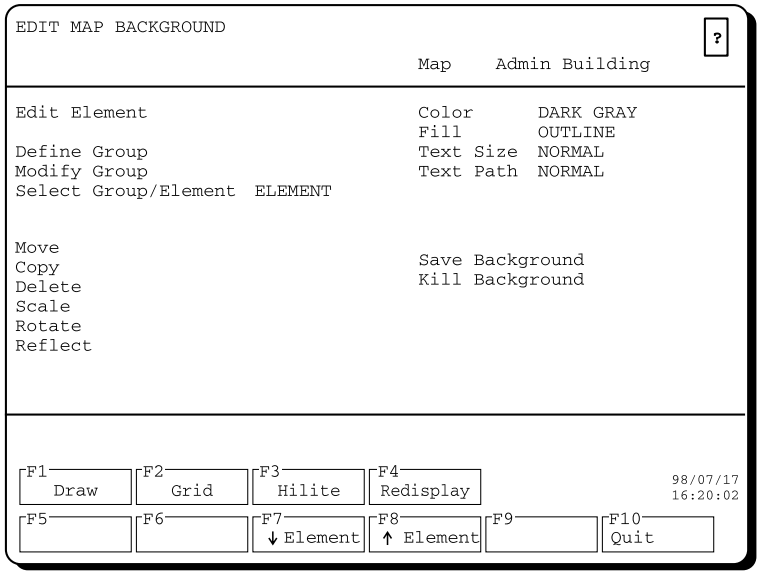
To use the same map background on a different map, you can copy the background. To copy the background to another map:

1. Press **F10** to exit from the Draw Map Background menu and return to the Define Site Maps menu.
2. Choose the **Select Map** function and pick the map that you want the background copied to. This becomes your current map.
3. Pick the **Copy Map Background** function and select the map whose background you want to copy from the list of maps. The selected map background copies to the current map.

If the current map already has a map background, it will be replaced with the copied background.

Editing the map background

To make any changes to the map, select **F1 - Edit**, from the Map Background menu. The screen displays the Edit menu.



The name of the current map is displayed in the Map field.

You can edit the map background by:

- redrawing some or all points on a selected object
- changing the position of the object on the screen
- changing the appearance of map objects by changing color, fill, text size or text path

Redrawing points on an object

To redraw an object on the primary screen:

1. Use the **F7** and **F8** keys to move around the objects on the primary screen, and highlight the object you want to edit. Each object is highlighted as you move around the screen.

2. Pick the **Edit Element** command. The cursor appears on the primary screen, positioned on the last point drawn on the highlighted object. For example, if the object selected is a rectangle, the cursor is positioned on the diagonal point of the rectangle.
3. Move the cursor to the next or previous point on the object with the **F7** (↓ Point) and **F8** (↑ Point) functions.
4. You're prompted to reposition the point. Move the cursor to a point on the screen, and press the **undo** key to erase the point. Then move to a new position on the screen and press the **pick** key to create a new point.

You must undo the point before repositioning it; otherwise, you'll just add another point to the object.

Changing the position of an object

The following functions are used to edit the position of an object on the map:

- move
- copy
- delete
- scale
- rotate
- reflect

Moving an object

To move an object to a new position on the map:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Move** function.

3. You're prompted to pick a reference point. Select any point on the object, e.g., a corner of a rectangle.
4. You're prompted to pick a destination point. This is the point that the reference point will be moved to. Move the cursor to where you want your object, and press the **pick** key. The object is positioned with the reference point at the destination point. If you want to adjust the destination point, move the cursor and press **pick** again. The destination point will move to the new position.

Copying an object

To make a copy of an object:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Copy** function.
3. You're prompted to pick a reference point. Select any point on the object, and press the **pick** key.
4. You're prompted to pick a destination point. Move the cursor to where you want the copied object to appear, and press the **pick** key. A copy of the object will appear, with the reference point of the copy positioned on the destination point. To make another copy, press the **pick** key again at another point.

Deleting an object

To delete an object from the map:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Delete** function. A pop-up menu appears, confirming that you want to delete the object.
3. Select **Yes** to delete the object, or **No** to cancel the function.

Once you delete an object you cannot recover it.

Rescaling an object

To rescale (resize) an object:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Scale** function.
3. You're prompted to pick an anchor point. This is the point that will be "anchored" in position when you rescale the object, and will prevent the object from moving. Select a point on the object (e.g., a corner of a rectangle), and press the **pick** key.
4. You're prompted to pick a reference point. This is the point that will move. For example, if you want to change the width of a rectangle, pick one of the top points. If you want to change the length, pick one of the bottom points.
5. You're prompted to pick a destination point. This is the point that the reference point will be moved to. Use the mouse or cursor keys to position the cursor and press the **pick** key. The reference point moves to the destination point, while the anchor point remains unchanged. To rescale the object again, press the **pick** key at another point.

Rotating an object

To rotate an object to a different angle:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Rotate** function.
3. You're prompted to pick the anchor point. This point will not move, i.e., the object is "anchored" at this point. Select a point on the object and press the **pick** key.
4. You're prompted to pick the angle. This is the number of degrees that you want the object rotated. Move the cursor up to increase the angle, and down to decrease the angle. The angle appears on the screen under the x and y coordinates (e.g., 45º).

5. Press the **pick** key to rotate the object. The object rotates to the specified angle. To rotate the object again, press the **pick** key again.

When you rotate a rectangle, it is considered to be a polygon.

Reflecting an object

To move an object about a horizontal or vertical plane:

1. Highlight the object by using the **F7** and **F8** keys.
2. Select the **Reflect** function.
3. You're prompted to pick the anchor point. This point will not move, i.e., the object will be *'anchored'* at this point. Choose a point on the object and press the **pick** key.
4. You're prompted to pick the plane. A vertical or horizontal line appears on the screen. Pressing any cursor key or moving the mouse toggles between vertical and horizontal. Press the **pick** key after selecting the plane. The object reflects to the specified plane, while remaining anchored at the anchor point.

Changing the appearance of map objects

You can change the fill, color, text size or text path options for any object on the map.

Selecting outline or fill

Rectangles, circles and polygons can be outlined or filled. If you specify outline, the object will be outlined in the color you select. If you specify fill, the object will be filled with the selected color.

You cannot fill lines or text.

The default setting is Outline. To change the default:

1. Use the **F7** and **F8** function keys to move around the objects on the primary screen, and highlight the object you want to change.
2. Select the **Fill** function, and choose Fill from the pop-up menu.

Selecting a color

To select a color for an object:

1. Use the **F7** and **F8** functions to highlight an object on the primary screen.
2. Select the **Color** function and choose a color from the pop-up menu. If you specified that your object be outlined, this will be the color of your outline. If you specified fill, the object will be filled with this color.

Selecting text size

To change the size of the text on the screen:

1. Use the **F7** and **F8** functions to highlight the text on the primary screen.
2. Select the **Text Size** function and choose an option from the pop-up menu.

Selecting text direction

To change the direction of the text on the screen:

1. Use the **F7** and **F8** functions to highlight the text on the primary screen.
2. Select the **Text Path** function and choose an option from the pop-up menu.

Editing a group of objects

The previous section dealt with editing individual objects on maps. You might want to perform some of these functions on a group of objects. The following functions can be performed on a group:

- move
- copy
- delete
- scale
- rotate
- reflect

You can't change the color, fill, text size or text path options for a group of objects. You can only make these changes on individual objects.

Combining objects in a group

To combine objects in a group:

1. Select the **Define Group** command. You're prompted to pick objects to be included in the group.
2. Use the **F7** and **F8** keys to move around the objects on the primary screen. As each object is highlighted, press the **pick** key to include it in the group. Press the **undo** key to exclude the object from the group.
3. When you've selected all the objects in a group, press the **done** key.

To add to or delete objects from the defined group:

1. Select the **Modify Group** function.
2. Use the **F7** and **F8** keys to move around the primary screen. Press the pick key to add another object to the group. Press the **undo** key to exclude an object from the group.
3. Press the **done** key to complete the function.

Performing functions on a group

To perform a function such as Move or Scale on a group, you must indicate that you want to work with the group.To select a group:

1. Choose the **Select Group/Element** function. This function is used to specify whether you're working with a group of objects, or an individual object.
2. Select the **Group** option from the pop-up menu.

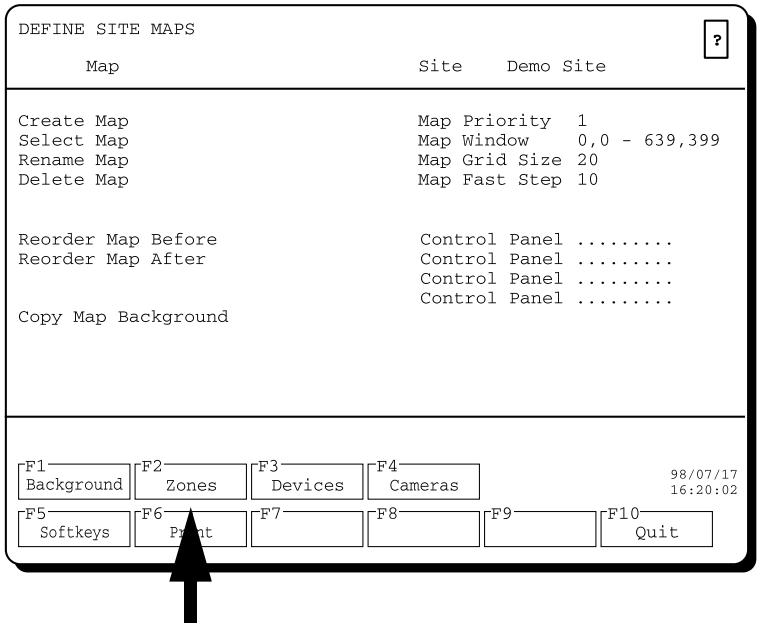
When you highlight the objects with **F3 - Hilite**, all the objects in the group are highlighted. You can then perform editing functions on the group as a whole.

To work with individual objects, pick the Select Group/Element function again, and select **Element** from the pop-up menu.

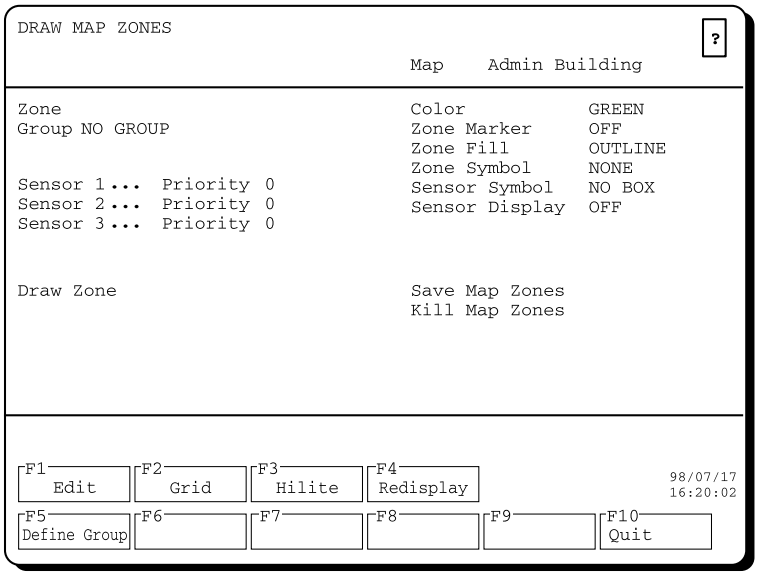
Entering the Zones menu

This section explains how to place zones and sensors on the map. A zone is an area covered by one to three sensors. You should have a hand-drawn map, indicating where the zones and sensors are. Use this hand-drawn map to place the zones and sensors on the screen map.

Select **F2 - Zones**, from the Define Site Maps menu.



The screen displays the Map Zones menu.



The **Map** field holds the name of your current map, selected in the previous menu.

In this menu you draw your map zones and sensors, and specify whether the zones are in a group.

Assigning zones, groups and sensors

For each zone on the map, you must assign a label, indicate whether it is part of a group, and list the sensors in the zone.

Assigning a zone lable

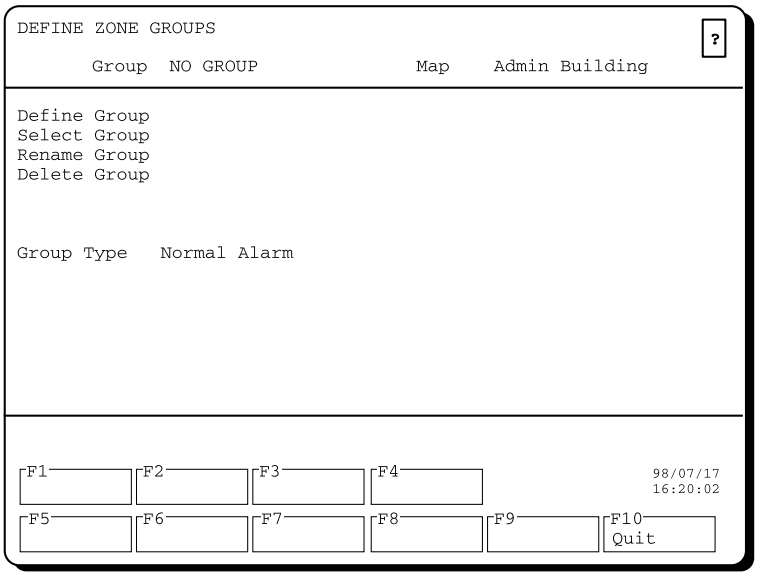
The **Zone** function is highlighted when you enter the menu. To create a new zone, press the **pick** key and enter a zone label (maximum 3 characters).

Assigning a zone to a group

Zones on a map can be grouped together based on a common element. This feature makes certain functions easier (e.g., you can access zones in a group in one step, or perform processing functions such as opening/ closing doors, turning lights on/off, etc.).

This group function is not the same as grouping objects on a map. Objects are grouped on a map only to make editing easier.

To create a new group, select **F5 - Define Group**, from the Draw Map Zones menu. The screen displays the Zone Groups menu.



The name of the current map is displayed in the **Map** field. The name of the current group (in this case, No Group) is displayed in the **Group** field.

Select the **Define Group** function and key in the group name (maximum 15 characters). This name automatically appears in the Group field.

Select the **Group Type** field and pick an option from the pop-up menu. Options are:

- normal alarm - sensor alarms that can be accessed and secured
- on/off control - objects (ie., lights) that can be turned on and off
- open/close control - objects (ie., doors) that can be opened and closed
- actv/deact control - objects that can be activated and deactivated

Press **F10** to return to the Map Zones menu.

Assigning a sensor to a zone

Each zone can contain one to three sensors. To list the sensors for the selected zone:

1. Select the **Sensor** function. Two input fields appear — one for the sensor label and one for the sensor priority.
2. Key in the sensor label.
3. Move to the next field and enter the sensor priority (a number from 0 to 64, with 64 being the highest priority).

The sensor priority is added to the map priority that you set earlier.

Specifying the appearance of zones and sensors

Once you have labeled the zones and related sensors, you can specify how the labels and symbols will appear on the map. The functions for specifying the appearance of zones and sensors are located on the right side of the menu. The defaults are set and you may want to keep these defaults.

Selecting a zone color

The zone color refers to the secure color — i.e., the color that indicates a non-alarm state. You have a choice of green or cyan (light blue). The default color is green. To change the default, select the **Color** function and pick cyan from the pop-up menu.

Selecting a zone symbol

The zone symbol indicates the area of detection for the zone. The area is represented by either a line or a polygon. This feature is especially helpful when the zone covers a certain section of the site perimeter — i.e., you can draw a line to specify the start and end points of that zone.

Select the **Zone Symbol** function and choose either Line or Polygon from the pop-up menu.

If you select a line as the zone symbol, you can specify whether the start and end points of the line are marked. Select the **Zone Marker** function, and choose **On** from the pop-up menu to have the markers appear on the map.

If you select a polygon as the zone symbol, you can specify whether the polygon is outlined or filled. Select the **Zone Fill** function, and choose **Outline** or **Fill** from the pop-up menu. The polygon will be outlined or filled with the zone color.

Selecting a sensor symbol

When the zone is displayed on the screen, the sensor label appears directly below the zone label. You can specify whether you want the sensor label to be enclosed in a symbol, and the type of symbol to display.

Select the **Sensor Symbol** function. Choose a symbol from the list, or pick **No Box** if you don't want any symbol for the sensor label.

The sensor label and symbol are always displayed on the map when the sensor is in an alarm or access state. To display the label and symbol when the sensor is secure, select the **Sensor Display** function. Choose **On** from the pop-up menu to have the label and symbol displayed. Choose **Off** to turn the display off (and display only the zone label).

Drawing zones on the map

After specifying the appearance of the zones and related sensors, you must draw the zones on the map.

Drawing zones with no zone symbol

If the zone does not have a zone symbol, follow the procedure below to draw the zone on the map.

1. Select the **Draw Zone** function.
2. You're prompted to pick the center point of the zone label. Use the mouse or cursor keys to position the cursor on the map, and press the **pick** key. The zone label appears on the screen, and the sensor label (and symbol, if assigned) appears underneath the label.

   If you have one sensor in the zone, the whole sensor label will be displayed. If you have more than one sensor in the zone, only the first letter of each label will be displayed.

3. You're prompted to reposition the zone, or end the command. To reposition the zone, press the **undo** key. Then pick a new position on the map. To leave the zone where it is, press the **done** key.

Drawing zones with a line symbol

If you specified a line symbol for the zone, follow the procedure below to draw the zone on the map.

1. Select the **Draw Zone** function.
2. You're prompted to pick the starting point of the line. Position the cursor where the zone begins, and press the **pick** key.

3. You're prompted to pick the end point of the line. Move the cursor to the end of the zone (a line is drawn on the screen as the cursor moves), and press the **pick** key.
4. The prompt "Enter end point" appears again. You can extend the line, move in another direction, etc. Press the **pick** key at the next end point.
5. When you've completed the line (i.e., picked the last end point), press the **done** key.
6. You're prompted to pick the center point of the zone label. Use the mouse or cursor keys to position the cursor on the map, and press the **pick** key. The zone label appears on the screen, and the sensor label (and symbol, if assigned) appears underneath the label.

> *If you have one sensor in the zone, the whole sensor label will be displayed. If you have more than one sensor in the zone, only the first letter of each label will be displayed.*

7. You're prompted to reposition the zone label, or end the command. To reposition the zone label, press the **undo** key. Then pick a new position on the map. To leave the zone label where it is, press the **done** key.

Drawing zones with a polygon symbol

If you specified a polygon symbol for the zone, follow the procedure below to draw the zone on the map.

1. Select the **Draw Zone** function.
2. You're prompted to pick the starting point of the polygon. Position the cursor where the zone begins, and press the **pick** key.
3. You're prompted to pick the vertex. Use the mouse or the cursor keys to position the cursor where you want the second point of the polygon to be (a line is drawn on the screen as the cursor moves), and press the **pick** key.

4. Continue the above procedure until you have the shape of the polygon you want.
5. When you've completed the polygon (i.e., picked the last point), press the **done** key.
6. You're prompted to pick the center point of the zone label. Use the mouse or cursor keys to position the cursor on the map, and press the **pick** key. The zone label appears on the screen, and the sensor label (and symbol, if assigned) appears underneath the label.

If you have one sensor in the zone, the whole sensor label will be displayed. If you have more than one sensor in the zone, only the first letter of each label will be displayed.

7. You're prompted to reposition the zone label, or end the command. To reposition the zone label, press the **undo** key. Then pick a new position on the map. To leave the zone label where it is, press the **done** key.

Drawing another zone

To draw another zone on the map:

1. Select the **Zone** field and key in the next zone number, overwriting the existing number.
2. If you want to specify another group for the zone, select the **Group** function and choose a new group from the pop-up menu.
3. Key in the list of sensors and sensor priorities.
4. To select different options for the new zone, move to the options column and choose the options you want.
5. Select the **Draw Zone** function and draw the zone on the map.

Saving the zones

Once you've drawn the map zones, ensure that you save them. If you try to exit from the Map Zones menu without saving your work, you'll be prompted to save any changes.

Select the **Save Map Zones** function, and pick **Yes** to save the zones. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

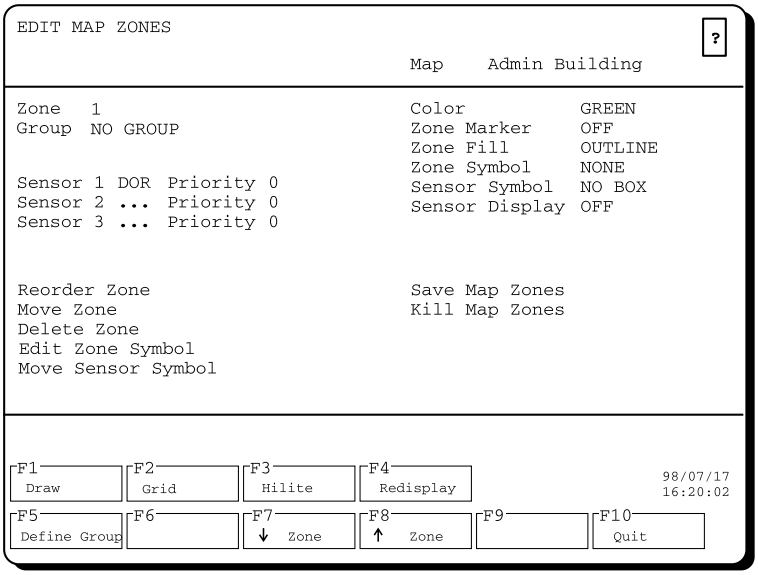
Deleting the zones

To delete all the map zones, select the **Kill Map Zones** function, and pick **Yes** from the pop-up menu.

CAUTION	*Once you delete the zones you cannot recover them. This function deletes all the zones even if you've previously saved them with the Save Map Zones command.*

Editing the zones

To make any changes to the zones, select **F1 - Edit**, from the Map Zones menu. The screen displays the Edit menu.



The name of the current map is displayed in the **Map** field. The name of the current zone is displayed in the **Zone** field.

You can edit the zones by:

- renaming the zones or sensors
- placing zones in another group, or modifying the group
- editing the position of the zone on the map
- editing the options associated with the zone

Use the **F7** and **F8** keys to highlight each zone on the map. When a zone is highlighted, the zone label, group and sensor data appear on the monochrome screen.

Renaming zones or sensors

To change a zone label:

1. Highlight the zone by using the **F7** and **F8** function keys. The zone label appears in the Zone field on the monochrome screen.
2. Select the **Zone** function and key in the new label, overwriting the existing one.

To change a sensor label:

1. Select the **Sensor** function for the sensor that you want to change. Key in a new sensor label, overwriting the existing one.
2. To change the priority, use the **Tab** key to move to the **Priority** input field, and overwrite the existing priority.

To add a new sensor to the zone:

1. Select the first blank sensor in the list and key in the new sensor label.
2. Move to the **Priority** field with the **Tab** key and enter a sensor priority in the input field.

Putting a zone in another group

To select another existing group for the zone, select the **Group** function, and choose another group from the pop-up menu.

To remove the zone from a group, choose **No Group** from the pop-up menu.

Modifying a group in the Define Group menu

To modify a group, press **F5 - Define Group**, to enter the Define Group menu. The current group is displayed in the Group field.

You can select another group to be the current group, rename the current group, delete the group, or choose another group type.

If the group has zones assigned to it, you'll get an error message when you try to delete the group. You can only delete a group if no zones have been assigned to it.

Editing a zone's position and shape

You can edit the size and shape of a zone symbol. You can reposition the zone and sensor label, and the zone (includes zone label/symbol and sensor label/symbol). You can also delete the zone.

Editing a zone symbol

To change the size or shape of a zone symbol:

1. Select the **Edit Zone Symbol** function. The cursor appears on the color screen, positioned at the last point drawn on the zone. If the zone symbol is a line, the cursor is positioned on the end point. If the zone symbol is a polygon, the cursor is positioned at the last point drawn.
2. Move the cursor to the next or previous points on the zone symbol with the **F7** (↓ Point) and **F8** (↑ Point) functions.
3. You're prompted to reposition the point. Move the cursor to a point on the zone symbol. Press the **undo** key to erase the point. Then move to a new position on the screen and press the **pick** key to create a new point.

You must undo the point before repositioning it; otherwise, you'll just add another point to the zone symbol.

Moving a zone label and sensor label/symbol

To reposition a zone label and sensor label and symbol on the map:

1. Select the **Move Sensor Symbol** function.
2. You're prompted to pick a reference point. Choose a point on the zone label, or the sensor label or symbol, and press the **pick** key.
3. You're prompted to pick a destination point. Move the cursor to where you want the label to be, and press the **pick** key. The zone label and the sensor label and symbol move so that the reference point is positioned on the destination point. To move the label again, move the cursor to another point and press the **pick** key.

The zone symbol does not move.

Moving a zone

To reposition an entire zone (zone label, sensor label and symbol, and zone symbol):

1. Select the **Move Zone** function.
2. You're prompted to pick a reference point. Choose a point on the zone and press the **pick** key.
3. You're prompted to pick a destination point. Move the cursor to where you want the zone to be, and press the **pick** key. The entire zone moves so that the reference point on the zone is positioned on the destination point. To move the zone again, position the cursor at a new point and press the **pick** key.

Deleting a zone

Deleting a zone involves deleting the zone label and symbol, and the sensor label and symbol.

To delete the current zone, select the **Delete Zone** function and pick **Yes** to delete the zone.

To delete all the zones from the map, use the **Kill Map Zones** function.

Reordering zones

When the operator steps through the zones on a map in the Operator menu, the zones are selected in the order that you have created them. You can change the order that the zones will be selected by using the Reorder Zone function.

To change the order of a zone in the sequence:

1. Use the **F7** and **F8** keys to highlight the zone that you want to change. The zone label and options appear on the monochrome screen.
2. Select the **Reorder Zone** function.
3. Highlight the zone that you want the original zone to be ordered after. Then press the **pick** key to reorder the zone.

> This function changes only the order in which the zones are selected and does not change the actual position of the zone on the map.

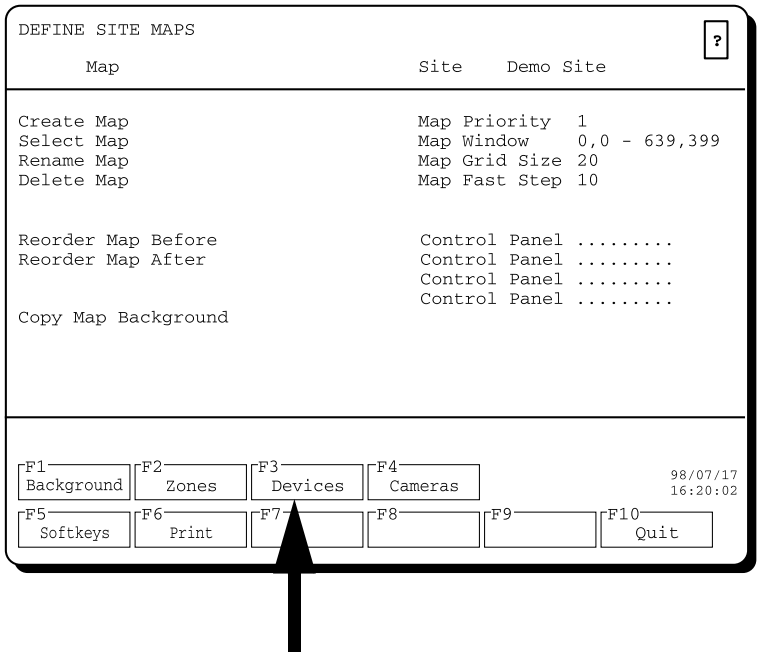
Editing the zone options

To edit any of the options associated with the zone - color, fill, symbol display - select the appropriate function and pick the option from the pop-up menu.

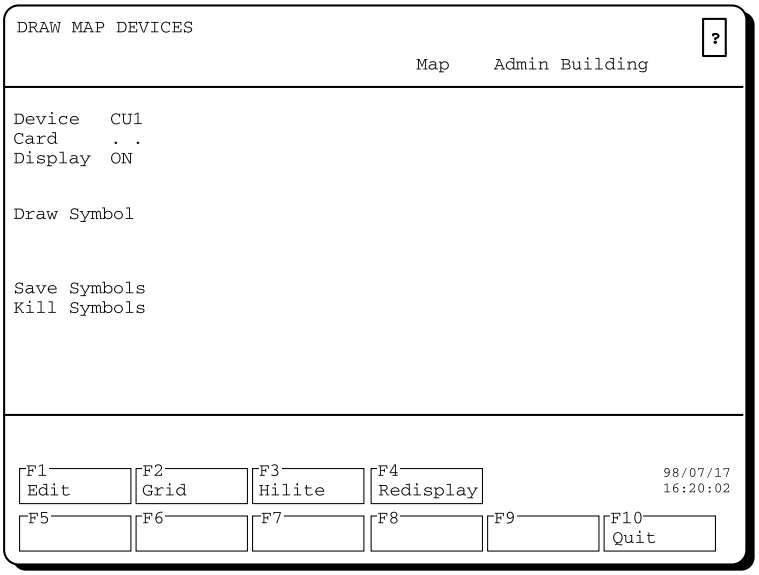
Entering the Devices menu

This section explains how to place the primary device and card symbols on the map. The symbols can appear on up to four site maps, as long as each device symbol appears somewhere on a map for the site. If you have only one map, all the device symbols must appear on that map.

Select **F3 - Devices**, from the Define Site Maps menu.



The screen displays the Map Devices menu.



The **Map** field holds the name of your current map, selected in the previous menu. The first device — CU1, is displayed in the **Device** field.

The **Card** field shows the card type for devices (CM, SN) that have cards.

The **Display** field indicates if the cards are to be displayed on the map during non-alarm conditions.

Drawing device and card symbols on the map

Device symbols appear as white, rectangular boxes on the screen. The 3 to 4-character device label appears inside each box. Device symbols and labels are generated automatically by the system, and cannot be changed.

Card symbols for the TM and TU may be displayed on the map. The symbol appears as a white rectangular box on the screen. The 3 to 4-character device label (CM# or SN#) and the 4-character card label (TM##, TU##, or CP##) appear inside the box. Card symbols and labels are generated automatically by the system and cannot be changed.

Drawing a device symbol

To draw a device symbol on the primary screen:

1. Select the **Draw Symbol** function.
2. You're prompted to pick the center point of the symbol. Position the cursor on the map, and press the **pick** key. It is recommended that you place the device symbols on an uncluttered portion of the screen, so that they can be clearly seen.
3. You're prompted to reposition the symbol, or end the command. To reposition the device symbol, press the **undo** key. Then pick a new position on the map. To leave the symbol where it is, press the **done** key.

Drawing another device symbol

To draw another device symbol on the map:

1. Select the **Device** function. A pop-up menu appears, listing all the primary devices you have in your system. The devices are in the order that you listed them in the Configure Hardware menu.
2. Select a device from the list.
3. Select the **Draw Symbol** function and draw the device on the color screen.

Drawing a card symbol

To draw a card symbol on the screen:

1. Select the **Device** field. A pop-up menu appears, listing all the primary devices you have in your system. Select the CM# or SN# device.
2. Select the **Card** field. A pop-up menu appears, listing all the available cards for the selected device. Select the card (TM##, TU##, or CP##).
3. Select the **Display** field. A pop-up menu appears. Select Off from the menu if you only want to display the card symbol during the alarm state. Select On if you want the card symbol to be displayed at all times.
4. Select the **Draw Symbol** function and follow the same steps as you would for drawing a device symbol.

Drawing another card symbol

To draw another card symbol on the map:

1. Select another card from the pop-up menu.
2. Set the display state to On or Off.
3. Select the **Draw Symbol** function to draw the card symbol.

Saving device and card symbols

Once you've drawn the device and/or card symbols, you should save them. If you try to exit from the Map Devices menu without saving your work, you'll be prompted to save.

Select the **Save Symbols** function and pick **Yes** to save the symbols. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

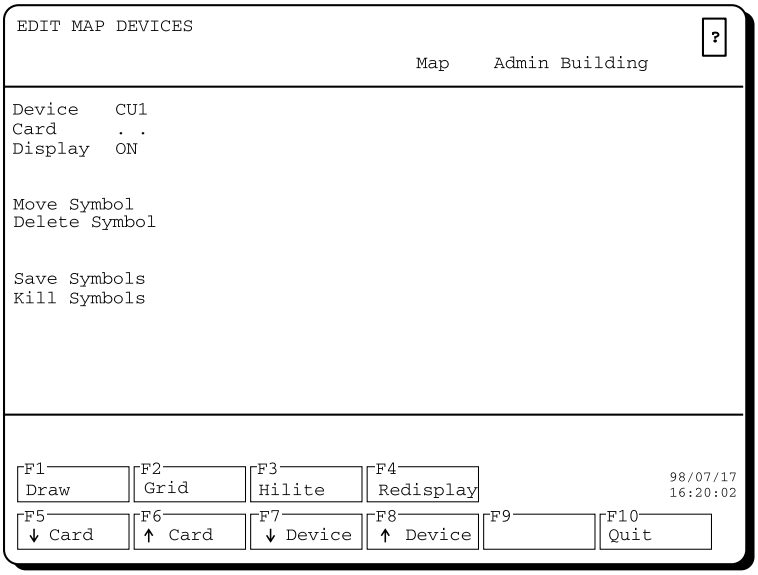
Deleting all symbols

To delete all the device and card symbols, select the **Kill Symbols** function and pick Yes from the pop-up menu.

CAUTION	Once you delete the symbols you cannot recover them. This function deletes all the symbols even if you've previously saved them with the Save Map Devices command.

Editing device and card symbols

To make any changes to the symbols, select **F1 - Edit**, from the Map Devices menu. The Edit menu appears.



The name of the current map is displayed in the **Map** field. The name of the last device symbol drawn is displayed in the **Device** field.

The Card type, if applicable, is shown in the **Card** field. The **Display** field shows the On/Off status.

You can edit the symbols by:

- moving the symbol to another position on the map
- deleting the symbol from the map

Use the **F7** and **F8** keys to highlight each device symbol on the map. When a device symbol is highlighted, the device name and display status appear on the monochrome screen.

Use the **F5** and **F6** keys to highlight each card symbol on the map for the selected device. When a card symbol is highlighted, the device and card names and display status appear on the monochrome screen.

Moving a symbol

To reposition a symbol on the map:

1. Select the **Move Symbol** function.
2. You're prompted to pick a reference point. Choose a point on the symbol and press the **pick** key.
3. You're prompted to pick a destination point. Move the cursor to where you want the symbol to be, and press the **pick** key. The device symbol moves so that the reference point on the symbol is positioned on the destination point. To pick another destination point, move the cursor to a new position and press the **pick** key again.

Deleting a symbol

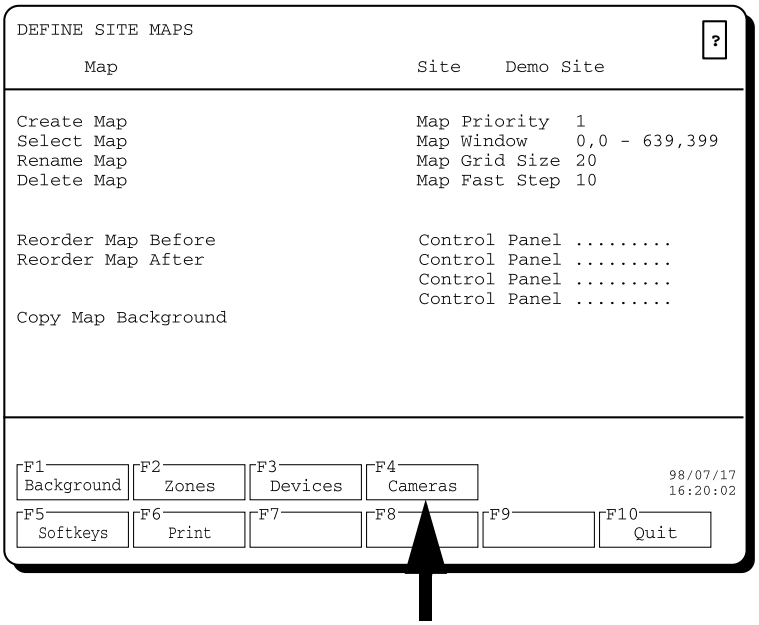
To delete the current symbol, select the **Delete Symbol** function and pick **Yes** from the pop-up menu.

To delete **all** the symbols from the map, use the Kill Symbols function.

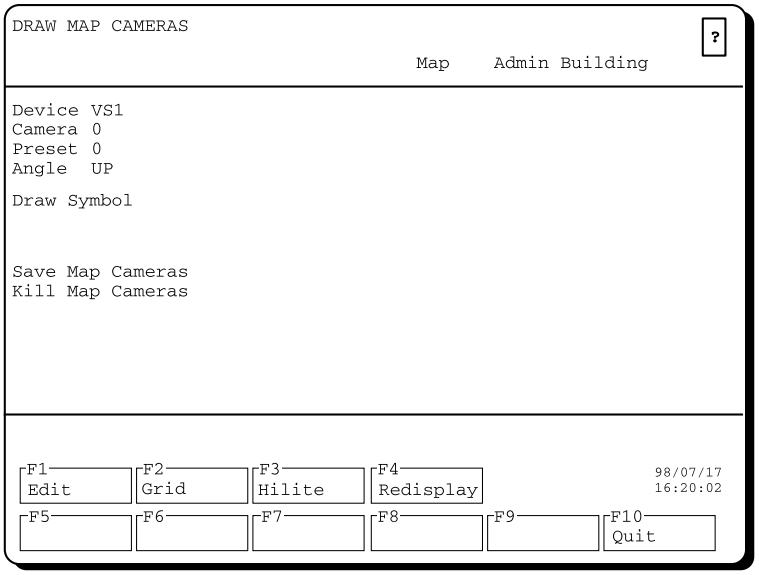
Entering the Cameras menu

This section explains how to place the camera symbols on the map. You must have a video switcher in your system in order to use this function.

Select **F4 - Cameras**, from the Define Site Maps menu.



The screen displays the Map Cameras menu.



Assigning camera numbers

To assign numbers to the cameras, select the **Camera** function and enter a number from 1 to 999 (maximum number of cameras). The default is 0, indicating that no cameras have been assigned to the current map.

Assigning camera presets

The preset function allows you to assign preset camera positions for the camera (if the video switcher you have supports this feature). Each camera can have up to 64 preset scenes.

Select the **Preset** function and enter a number from 1 to 8. The default is 0, indicating that no presets have been assigned to the current camera.

Setting camera angles

You can set the angle of the camera symbol to display how a sensor is being monitored. To set the camera angle, select the **Angle** function and choose the angle you want from the pop-up menu.

Drawing camera symbols on the map

Camera symbols appear as white arrows on the screen, pointing in the direction that you set in the **Angle** field. The camera number and preset number also appear.

Drawing a camera symbol

To draw a camera symbol on the primary screen:

1. Select the **Draw Symbol** function.
2. You're prompted to pick the position of the symbol. Position the cursor on the map, and press the **pick** key. The camera symbol appears, pointing in the direction that you set in the Angle field (Up, Down, etc.). The camera and preset number appears in the direction that the camera is pointed.
3. To reposition the camera symbol, press the **undo** key, then pick a new position on the map. To leave the symbol where it is, press the **done** key.

Each camera symbol will eventually have a monitor number associated with it. This monitor number is assigned in the point assignment menu. (See chapter 6, Assigning input and output points). The camera and preset number does not appear when the camera is displayed on final system, but the monitor number associated with it will appear in the open end of the symbol.

Drawing another camera symbol

To draw another camera symbol on the map:

1. Select the **Camera** function and key in the next camera number, overwriting the existing number.
2. Select the **Angle** function, and choose an angle from the pop-up menu.
3. Select the **Draw Symbol** function, and draw the camera symbol on the primary screen.

Saving camera symbols

Once you've drawn the camera symbols, ensure that you save them. If you try to exit from the Map Cameras menu without saving your work, you'll be prompted to save.

Select the **Save Map Cameras** function and pick **Yes** to save the symbols. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

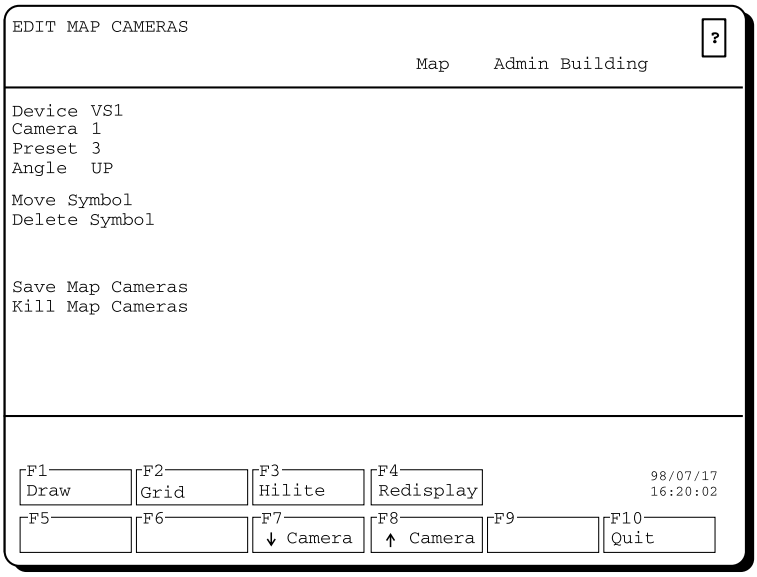
Deleting all camera symbols

To delete all the camera symbols, select the **Kill Map Cameras** function and pick **Yes** from the pop-up menu.

CAUTION	Once you delete the camera symbols you cannot recover them. This function deletes all the camera symbols even if you've previously saved them with the Save Map Cameras command.

Editing the camera symbols

To make any changes to the camera symbols, select **F1 - Edit**, from the Map Cameras menu. The screen displays the Edit menu.



The name of the current map is displayed in the **Map** field. The number of the last camera symbol drawn is displayed in the **Camera** field.

You can edit the camera symbols by:

- renumbering the cameras and presets
- changing the camera angle
- moving the symbol to another position on the map
- deleting the symbol from the map

Use the **F7** and **F8** keys to highlight each camera symbol on the map. When a camera symbol is highlighted, the camera number appears on the secondary screen.

Renumbering cameras

To change the current camera number, select the **Camera** function and enter the new number in the input field, overwriting the existing one.

If you give the camera a number that is being used for another camera, you'll get an error message.

To interchange two camera numbers:

1. Renumber the first camera (e.g., camera 1) to a camera number that has not been assigned (e.g., renumber camera 1 to camera 100).
2. Renumber the second camera (e.g., camera 2) to the first camera number (e.g., renumber camera 2 to camera 1).
3. Renumber the first camera to the second camera number (e.g., renumber camera 100 [previously camera 1] to camera 2).

To change the current camera preset, select the **Preset** function and enter the new number in the input field, overwriting the existing one.

Changing a camera angle

To change a camera angle, select the **Angle** function and pick another angle from the pop-up menu. The camera angle changes on the primary screen.

Moving a camera symbol

To reposition a camera symbol on the map:

1. Select the **Move Symbol** function.
2. You're prompted to pick a reference point. Choose a point on the symbol and press the **pick** key.

3. You're prompted to pick a destination point. Move the cursor to where you want the symbol to be, and press the **pick** key. The camera symbol moves so that the reference point on the symbol is positioned on the destination point. To move the symbol to another point, position the cursor at another point and press the **pick** key.

Deleting a camera symbol

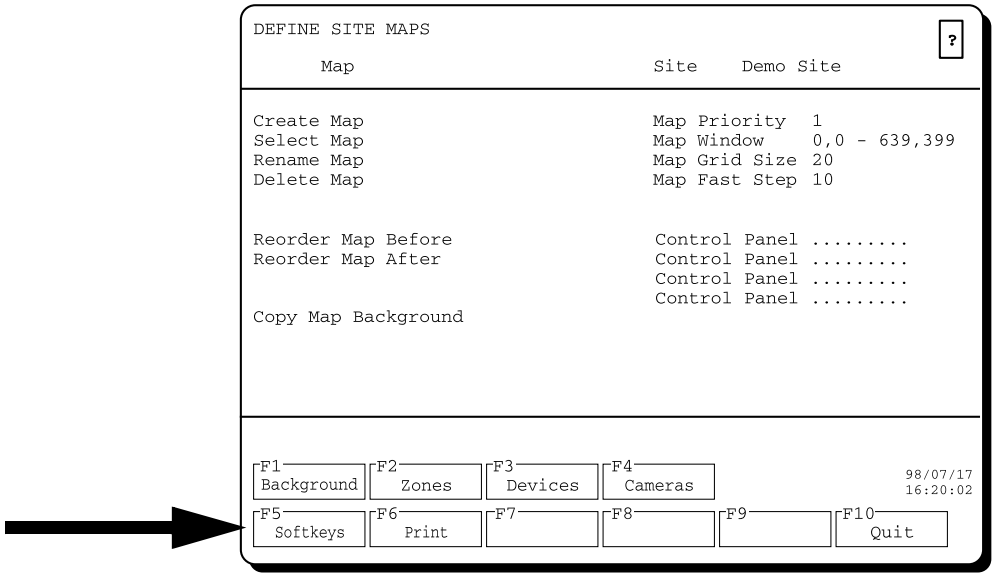
To delete the current camera symbol, select the **Delete Symbol** function and pick **Yes** to delete the symbol.

To delete all the symbols from the map, use the **Kill Map Cameras** function.

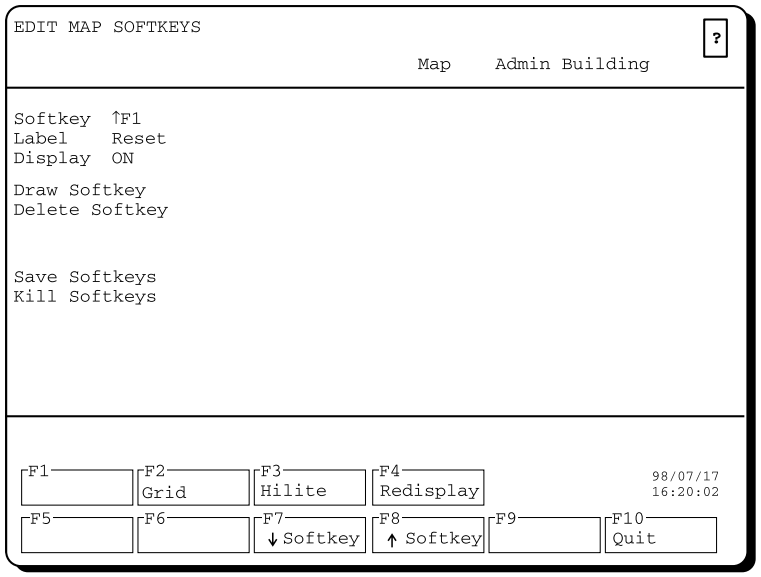
Entering the Softkeys menu

This section explains how to place the softkeys on the map. A softkey is a key symbol that appears and is active on the map. a maximum of 4 softkeys can appear on a map.

Select **F5 - Softkeys** from the Define Site Maps menu.



The screen displays the Edit Map Softkeys menu.



Softkeys appear as rectangles with a double outline. A label appears inside each softkey. The size of the softkey is set by the system.

Creating a softkey

To create a softkey on the map:

1. Select the **Softkey** field. A pop-up menu appears listing the available softkeys. Select the softkey designation ↑**F1** to ↑**F4.**
2. Select the **Label** field. Enter the name of the softkey up to a maximum of 20 characters.
3. Select the **Display** field. Select **ON** or **OFF**.
4. Select the **Draw Softkey** function.
5. You're prompted to pick the position of the softkey. Position the cursor on the map, and press the **pick** key. The softkey symbol appears.
6. To reposition the softkey symbol, press the **undo** key, then pick a new position on the map. To leave the symbol where it is, press the **done** key.

Repeat this procedure for each softkey.

Saving softkeys

Once you've drawn the softkeys, ensure that you save them. If you try to exit from the Map Softkeys menu without saving your work, you'll be prompted to save.

Select the **Save Softkeys** function and pick **Yes** to save the softkeys. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

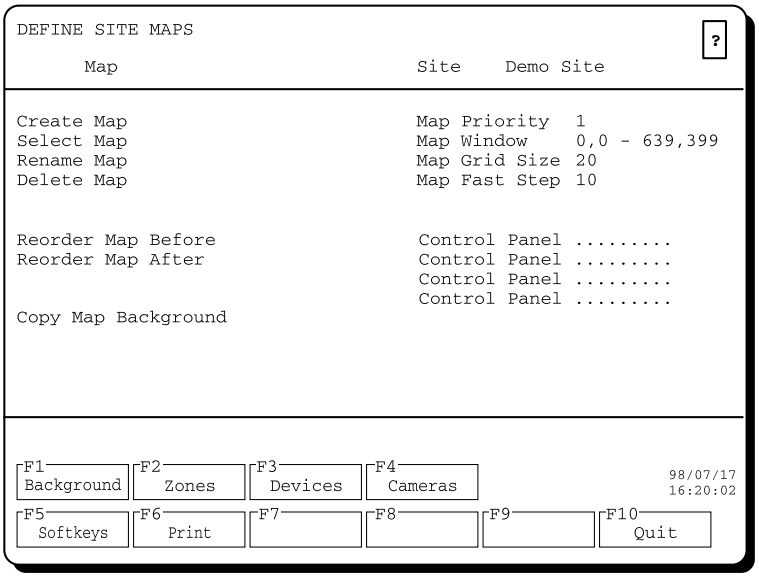
Deleting all softkeys

To delete all the softkeys, select the **Kill Softkeys** function and pick **Yes** from the pop-up menu.

CAUTION	Once you delete the softkeys you cannot recover them. This function deletes all the softkeys even if you've previously saved them with the Save Softkeys command.

Printing the map

To print the map displayed on the color monitor, select **F5 - Print**, from the Define Site Maps menu.



Maps can be printed only on 9 or 24-pin printers that support EPSON bit graphics commands.

5 Creating datalog and remote alarms

Overview

When you planned out your site you had the option of having some alarms print on the event logger. These alarms are ones that you would want to track but that don't require the operator's immediate attention, for example, a guard tour, or a VCR reaching the end of a tape. The alarms are referred to as **datalog** alarms.

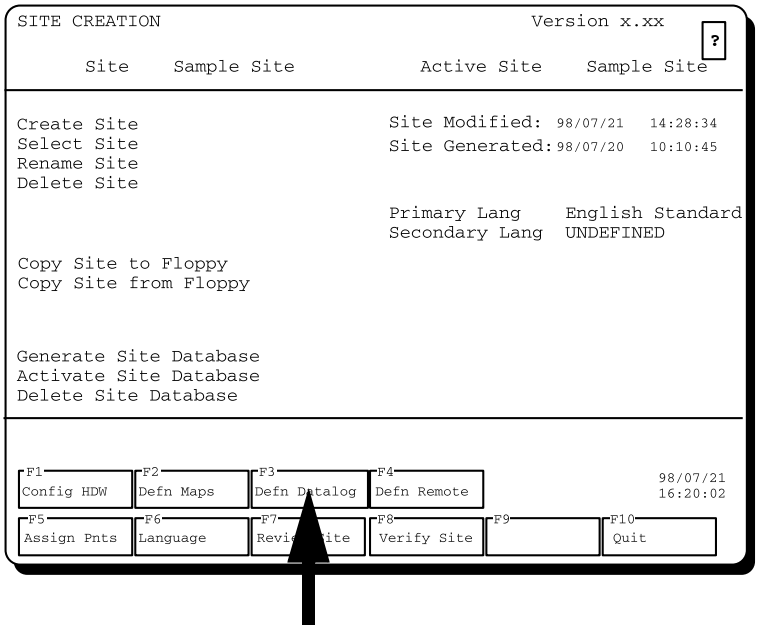
Your site might also include some alarms that are annunciated at a remote location. These alarms are referred to as **remote** alarms.

You must identify each datalog and remote alarm that you want the system to monitor. A datalog alarm is identified by a unique type, location and on/off state. A remote alarm is identified by a unique type and location.

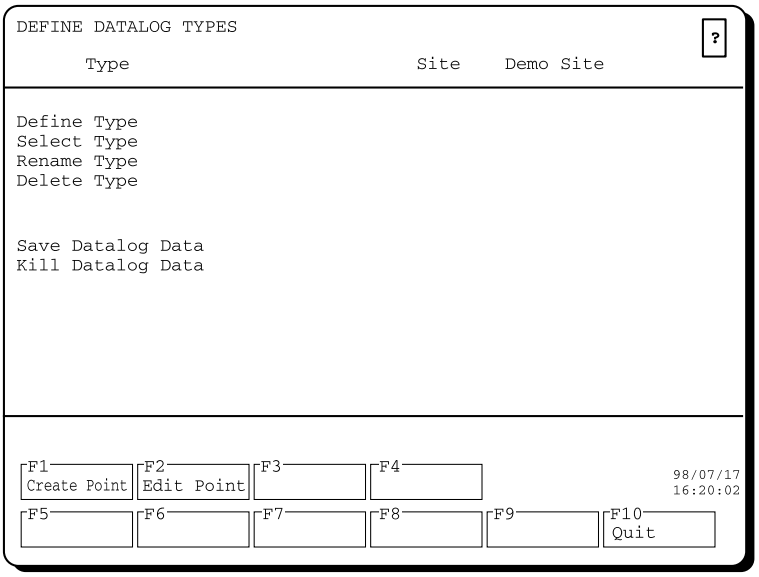
This chapter explains how to enter the information in the site database for the datalog and remote alarms in your system. The procedure is the same for both types of alarms; therefore, a detailed explanation is given only for the datalog alarms.

Entering the Defn Datalog menu

To enter the identifying data for datalog alarms, select **F3 - Defn Datalog** from the Site Creation main menu.



The screen displays the Datalog Types menu.



The **Site** field holds the name of the site database that you created in the main menu.

The **Type** field is blank because no datalog types have been created yet.

Specifying a datalog type

To create a datalog alarm, you must first specify the datalog type. A datalog type is a general way of grouping datalog alarms.

To enter the datalog type, select the **Define Type** function and key in the type name, e.g., VCR.

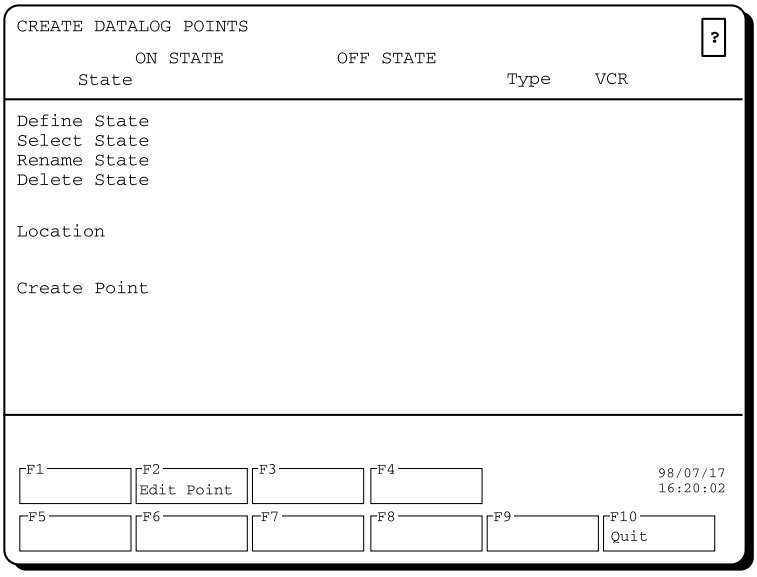
Editing datalog types

You can rename a datalog type at any point. You can delete a datalog type **only** if you haven't yet specified a location and on/off state or have deleted all locations for the datalog type.

Specifying a location and on/off state

After you've identified a datalog type, you must specify a location and on/off state for the alarm.

Select the **F1 - Create Point** function. The screen displays the Datalog Points menu.



The current datalog type is displayed in the **Type** field. The **State** field is blank because no states have been created yet.

Specifying an on/off state

To specify on and off states for a datalog alarm:

1. Select the **Define State** function. Two input fields appear Ð one for the On state and one for the Off state.
2. Key in the On state (e.g., 'VCR on').
3. Press the **Tab** key to move to the next input field and key in the Off state (e.g., 'End of tape').
4. Press the **Enter** key to complete the function. The state information appears in the State field.

Editing an on/off state

The following functions can be used to make changes to the on/off states.

Selecting another state

To choose another on/off state, choose the **Select State** function. A pop-up menu appears that contains a list of all the on/off states that have been created. Select another on/off state from the list.

If no states have been created, you'll get an error message when you pick this function.

Renaming a state

To rename the current on/off state, choose the **Rename State** function and enter the new on/off states in the input fields.

> If you want to change only the on or off state, you must still re-key both states.

If you rename the on/off state to an existing on/off state, you'll get an error message.

Deleting a state

To delete the current on/off state, select the **Delete State** function.

> Once you've created a datalog point with this state, you cannot delete the state without deleting all points using that state. See Creating a datalog point, below.

Specifying a location

To specify a location for a datalog alarm, select the **Location** function and key in the location (e.g., VCR1).

Creating a datalog point

When you've specified a state and location, you must consolidate this information into one unique datalog alarm. Select the **Create Point** command.

The system automatically creates a point with the specified location and on/off states. This point is not a physical point on a hardware device, but is the system's way of storing the alarm information.

When the point has been created, the message `"Datalog point created!"` appears on the prompt line.

Once you've created the point, you can't delete the state or the type. You **can** rename the state or type.

Creating another datalog alarm

A datalog alarm is made up of a unique type, location and on/off state. As long as one of the three is different for each alarm, the alarm is considered unique. You can therefore have several states for one location, or several locations with similar states.

To create another datalog alarm, identify a new state or location, or both.

To change the state, select the **Define State** function and key in a new state. Or pick the **Select State** function, and choose another state from the pop-up menu. Then select **Create Point**.

To change the location, select the **Location** function and key in a new location. Then select **Create Point**.

If you try to create another alarm without changing either the state or location, you'll get an error message that you're trying to create a duplicate alarm.

Saving datalog alarms

Once you've created all the datalog alarms, ensure that you save them. If you try to exit from the Define Datalog Types menu without saving your work, you'll be prompted to save.

Select the **Save Datalog Data** function. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

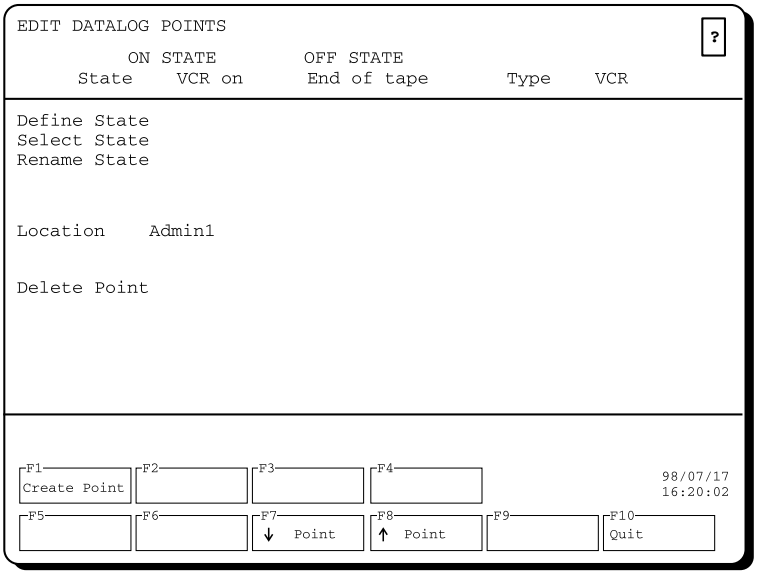
Deleting all datalog alarms

To delete all the datalog alarms, select the **Kill Datalog Data** function.

CAUTION	Once you delete the datalog alarms you cannot recover them. This function deletes all the datalog alarms even if you've previously saved them with the Save Datalog Data command.

Editing datalog alarms

To modify the datalog alarms, select **F2 - Edit Point**. The screen displays the Edit menu.



The current state and type are displayed in the **State** and **Type** fields. To edit the alarms for another type, return to the Define Datalog Types menu and choose another type with the Select Type function.

You can edit the datalog alarms by:

- renaming the on/off state
- renaming the location
- deleting the alarm

Use the **F7** and **F8** keys to select another datalog alarm to edit. The alarm information appears in the State and Location fields.

Renaming an on/off state

You can rename an on/off state by:

- creating a new state that becomes the current state
- modifying the current state
- selecting a new state from a list of created states

Creating a new state

To create a new on/off state for an alarm, choose the **Define State** function and key in the new on and off states.

When you return to the Create Datalog Points menu, the old state still appears in the State field, and in the list of selectable states. However, an alarm for that state no longer exists. The information remains on the screen so that you can create a new alarm using the existing states.

Modifying a state

To modify the current state, select the **Rename State** function and key in the on and off states. Remember to enter both states, even if you're only renaming one.

This function works the same way as the Rename State function in the Create Datalog Points menu.

Selecting a state from a list of created states

To select a new state from a list of created states, choose the **Select State** function and choose another state from the pop-up menu.

This function is different from the Select State function in the Create Datalog Points menu, in that there you must create a new alarm (with the Create Point function) once you've selected a new state. In the Edit menu, the selected state automatically replaces the current one.

Renaming a location

To rename a location, select the **Location** function and key in the new location in the input field.

Deleting an alarm

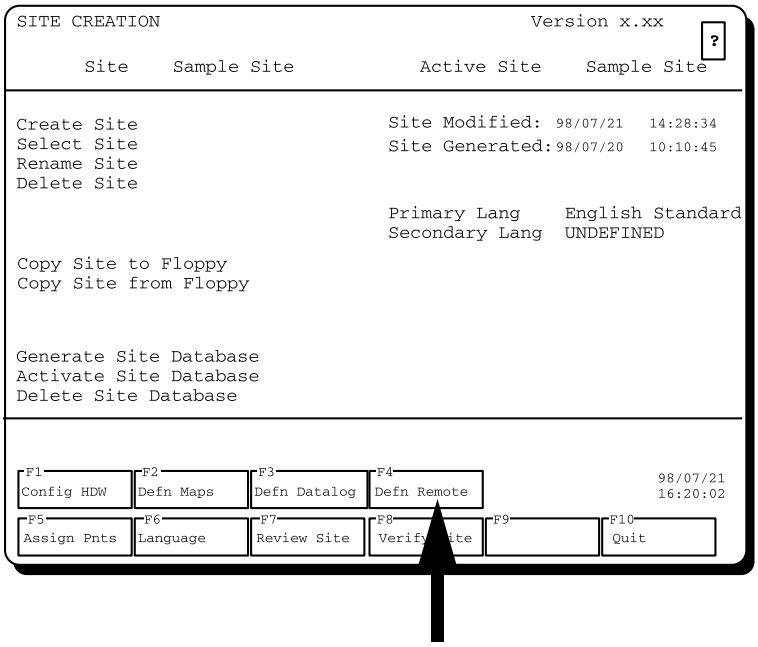
To delete the current alarm, select the **Delete Point** function.

If only one alarm has been created for the current type, the state and location fields become blank. If more than one alarm exists for that type, the state and location fields show the next alarm information.

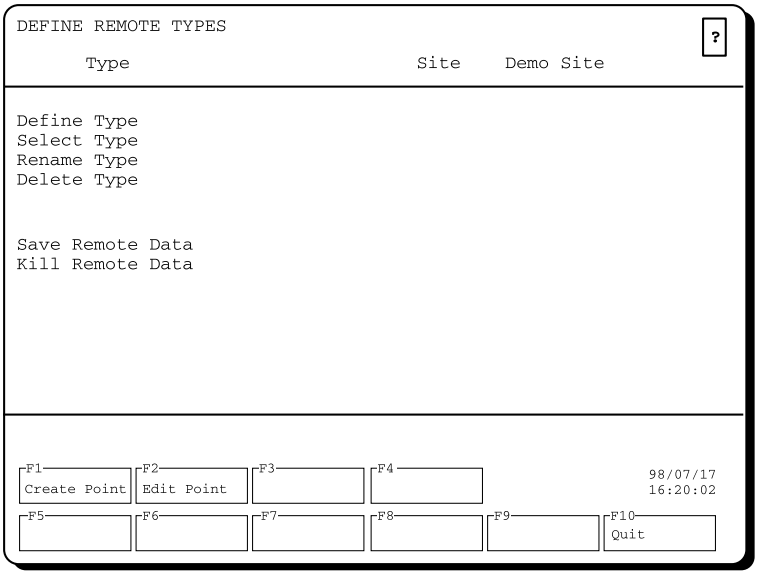
If you return to the Create Datalog Points menu (F1 - Create Point), you'll see that the deleted information is still displayed. However, the alarm does not exist. The information remains on the screen so that you can create a new alarm by editing the existing data.

Creating remote alarms

To enter the identifying data for remote alarms, select **F4 - Defn Remote** from the Site Creation menu.



The screen displays the Remote Types menu.



The **Site** field holds the name of the site database that you created in the main menu.

The **Type** field is blank because you haven't created any remote alarm types yet.

To create a remote alarm you must specify the remote type and a location for the alarm (e.g., type is Door Alarm, location is Door 1). This procedure is the same as the one for creating datalog alarms.

6 Assigning input and output points

Overview

For each alarm in your system, you must specify the input and output point assignments. The point assignments are the links between the map zone display and the physical points on the hardware devices.

The types of alarms are listed in the table below.

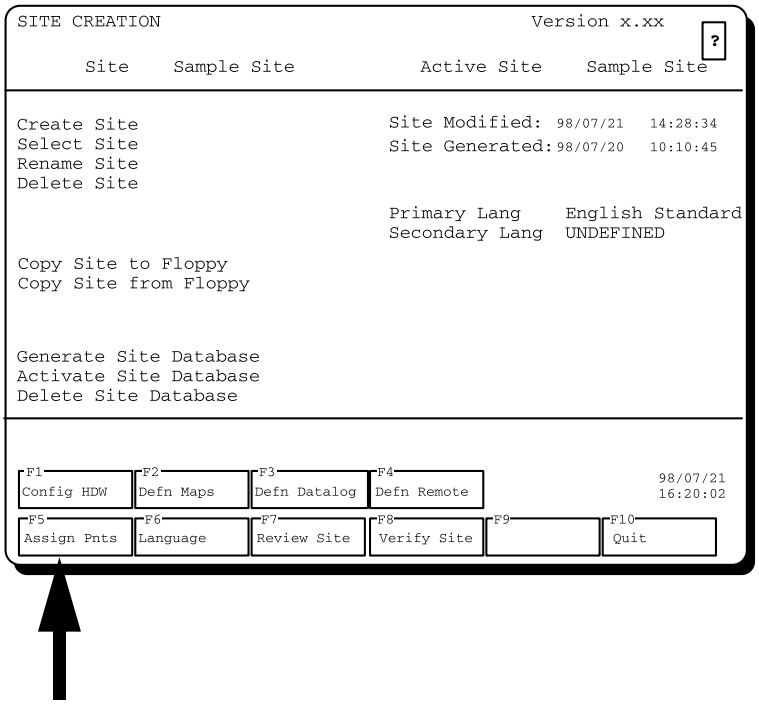
Alarm	Description
Display	alarms that are displayed on the site maps when a sensor is activated
Diagnostic	alarms that are displayed on the site maps when a hardware device or card fails
Datalog	alarms that are printed on the event logger when the associated device changes state
Remote	alarms that are annunciated at a remote location when a sensor is activated
Duress	alarms that are annunciated at a remote location when an operator enters a duress code at the control unit, unsuccessfully attempts to enter a password, or fails to process an alarm.

This chapter gives you instructions on entering the point assignment data in the database. The procedure is similar for all alarms; therefore, a detailed explanation is given only for display alarms.

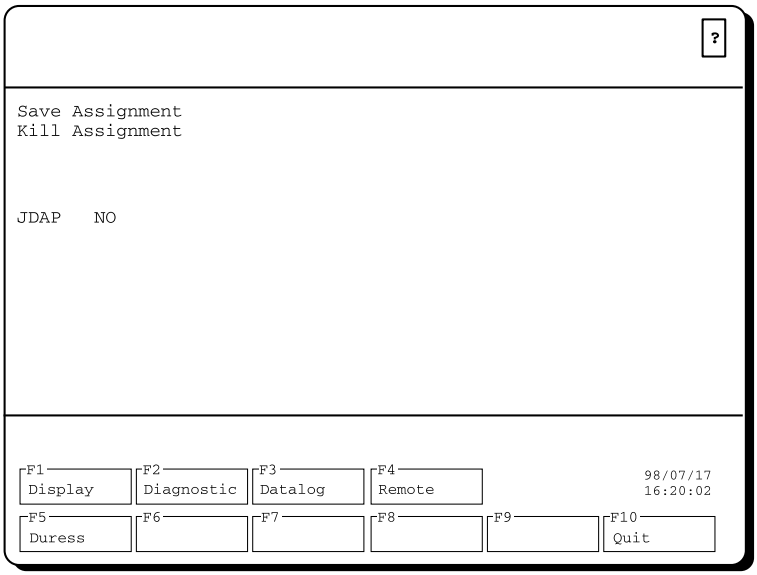
Datalog, remote and duress alarms are optional. If you are not monitoring these alarms in your system, you can ignore the sections that deal with these alarms.

Entering the Assign Pnts menu

To assign points to alarms in your system, select **F5 - Assign Pnts** from the Site Creation main menu.



The screen displays the Assign Points menu.

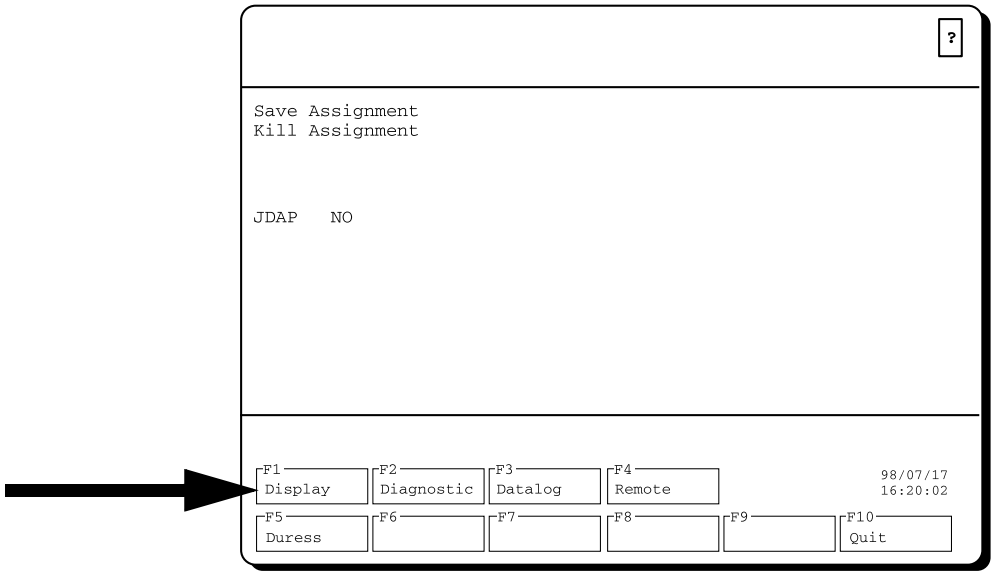


The five types of alarms are accessed by the function keys.

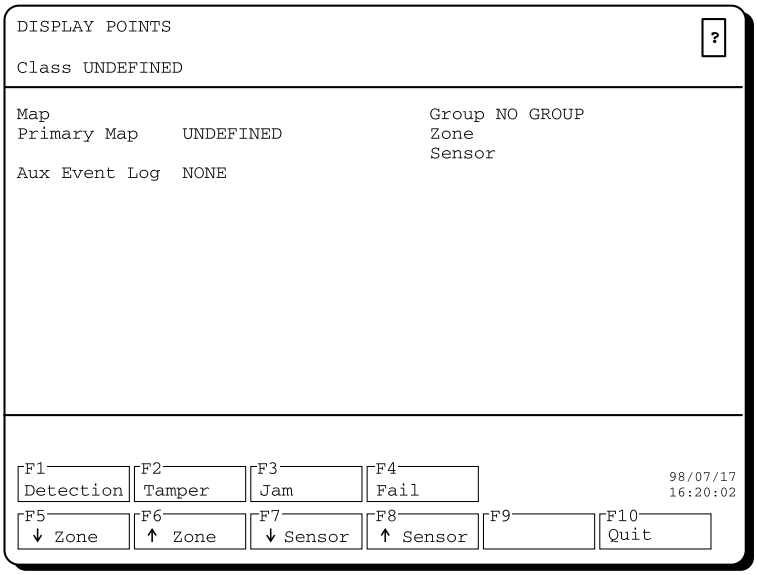
If you are using the Enhanced Alarm Processing feature, select JDAP (Joint Domain Alarm Processing), and choose **Yes** from the pop-up menu. Enhanced Alarm Processing can be used with display and remote alarms.

Assigning points for display alarms

To assign points for the display alarms, choose **F1 - Display** from the Assign Points menu.



The screen shows the Display Points menu.



The possible sub-types of alarms (detection, tamper, jam and fail) are accessed by the function keys.

Selecting a display alarm

To assign points to a display alarm, you must first select the alarm.

1. Choose the **Map** function and select a map from the pop-up menu. The map appears on the primary screen. The system automatically chooses the first group, zone and sensor, and highlights them on the primary screen. As well, the group, zone and sensor names are displayed in the corresponding fields on the secondary screen.
2. To select another zone, use the **F5** and **F6** keys to move around all the zones on the displayed map. As you move around the map, each zone is highlighted on your map and the corresponding zone label appears on the secondary screen.
3. To select another sensor, use the **F7** and **F8** keys to move around all the sensors on the selected zone.

Assigning maps as primary and secondary

A sensor can appear on more than one map (e.g., if the maps have overlapping areas), as long as the sensor, zone and group labels are the same on both. If the sensor you've chosen appears on more than one map, you must assign one map as the primary map and the others as secondary maps. When an alarm is generated for that sensor, the map that has been designated as the primary map is the one that's displayed for the operator.

When you select a map, the **Class** field displays "Primary".To make the current map a secondary map, select the **Primary Map** function and choose the map you want as the primary map.

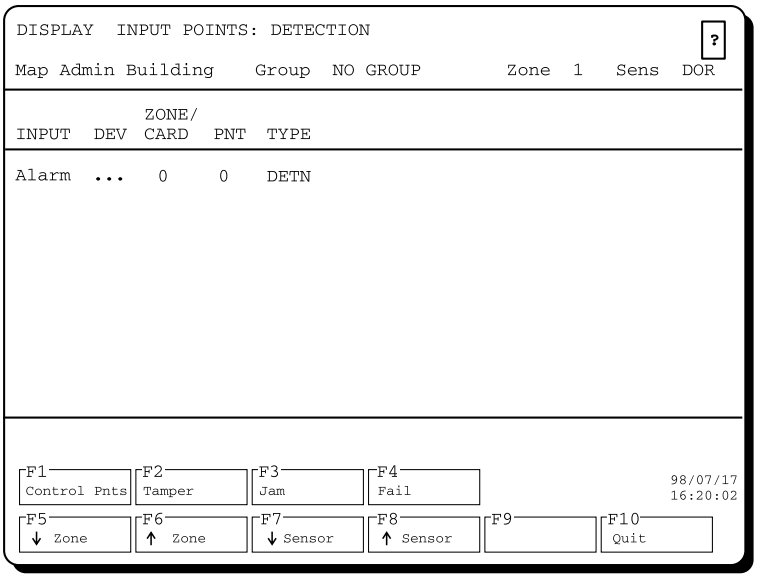
The map name is displayed in the Primary Map field, and the current map's class changes to Secondary. This map is only considered secondary for this particular display alarm. For any display alarm that appears only on this map, the map is considered primary.

If the map you select does not contain the display alarm, you'll get an error message.

Once you've chosen the display alarm, you must assign points for each sub-type of alarm (detection, tamper, jam and fail) generated by the display alarm.

Assigning input points

Choose **F1 - Detection** from the Display Points menu to assign points for detection alarms. If you are **not** using the Enhanced Alarm Processing feature, the screen displays the following menu.



In this menu you assign the input point for the detection alarm.

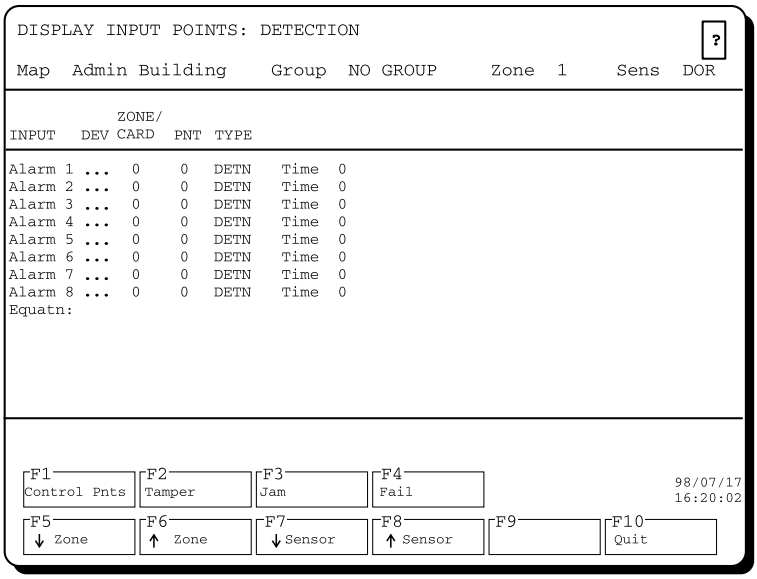
An input point for an alarm is a physical point on a hardware device, that collects data from the associated alarm. When activated, the input point causes the alarm to change state.

Select the alarm input and enter the device name, TM zone or card number, point number, and input type.

In the **Dev** field, use the 3 to 4-character label (e.g., IU1). In the **Type** field, enter the first letter of the type (detection, tamper, jam or fail).

Assigning input points with Enhanced Alarm Processing

If you are using the Enhanced Alarm Processing feature, the following screen appears when you choose **F1 - Detection** from the Display Points menu.



With the enhanced alarm processing option, you can configure an alarm state as a combination of several inputs (maximum eight).

Combining alarms with no time element

To combine alarms without using a time factor:

1. Fill in the input data (input device, zone/card number, point number, and type of input point).
2. Move to the **Equation** field and enter an equation for combining the alarms. Combine alarms using **& (and)** or **| (or)**. For example:
   - 1 & 2 (spaces optional) alarm 1 and alarm 2 must be active at the same time
   - 1 & 2 | 3 alarm 1 and alarm 2 must be active, or alarm 3 must be active
   - 1 & (2| 3) alarm 1 and alarm 2 must be active, or alarm 1 and alarm 3 must be active

Combining alarms with a time element

To combine alarms with a time factor:

1. Fill in the input data (input device, zone/card number, point number, and type of input point).
2. Move to the **Time** field and enter a time period from 0 to 60 seconds.

   The time option sets the active time of the alarm. For example, setting the Time to 30 for an input point means that when an alarm on that point is deactivated, it is considered to be active for an additional 30 seconds.
3. Move to the F5 - Assign Pnts field and enter an equation for combining the alarms. Combine alarms using **& (and)** or **| (or)**. For example:
   - 1 & 2 — alarm 2 must occur within 30 seconds of alarm 1 ending

Assigning control points

A control point is a physical point on a hardware device that, when activated, changes the state of the display alarm.

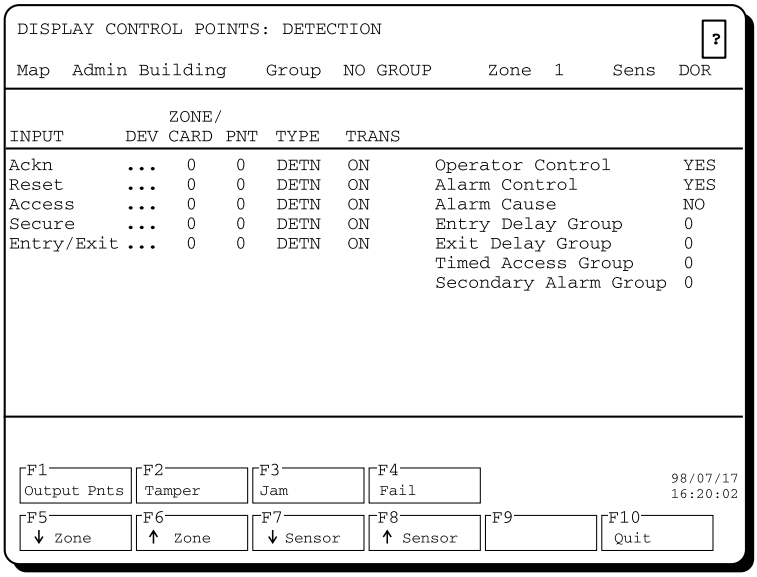
All sub-types of display alarms can have the following state changes:

State	Description
Acknowledge	• a transition on the input point acknowledges the alarm
Reset	• a transition on the input point resets the alarm
Access	• a transition on the input point places the alarm into access
Secure	• a transition on the input point places the alarm in a secure state
Entry/Exit	• a transition on the input point places the alarm into the access state from a secure or entry delay state, if the alarm does not have a defined time access delay • a transition on the input point places the alarm into the timed access state from a secure or entry delay state, if the alarm has a defined timed access delay • a transition on the input point places the alarm into the exit delay state from an access or timed access state

For each state that has configured points, you must record the input device, TM zone or card number, point number, type of input point configured for it, and the transition (Off to On [normal to abnormal state], or On to Off [abnormal to normal state]).

You do not need to have points configured for each state.

To enter control points, press **F1 - Control Pnts** from the Input Pnts menu. The screen displays the Control Points menu.



In this menu, you:

- enter the point configuration for the control points
- set the processing parameters for the display point

To enter the point configuration for a state, select the input and enter the device name, TM zone or card number, point number, input type and transition.

In the **Dev** field, enter the 3 to 4-character device code.

In the **Type** field, enter the first letter of the type (detection, tamper, jam or fail).

In the **Trans** field, enter On for an Off to On transition, and enter Off for an On to Off transition.

For each display point, you can also set the processing parameters such as Operator access, Alarm secure, Alarm causes, Entry delay group, Exit delay group, Timed access group, and Secondary alarm group.

Operator control

This defines the options that are available to the operator for disabling (accessing) the alarm annunciation of individual zones.

If **Secure** is selected, then the Access function key is not available when viewing sensors with the Acc/Sec function during normal operator processing. However, the operator **is** able to access the sensor during alarm processing. Select the **Operator Control** function and choose **Secure** or **Acc & Secure** from the pop-up menu.

Alarm control

You can specify whether the operator can perform Access, Reset or Both operations on detection alarms during alarm processing. If the Reset option is disabled, the zone can be reset/secured only during normal operator processing or through an input control point. Select the **Alarm Control** function and choose **Access**, **Reset** or **Acc & Reset** from the pop-up menu.

Alarm cause

You can specify whether the operator must select an alarm cause as part of alarm processing for each alarm. If you specify Yes and the supervisor has created a list of alarm causes, the list appears after the operator has

accessed, secured or released the alarm. The operator must select an alarm cause in order to finish processing the alarm. Select the **Alarm Cause** function and choose **Yes** or **No** from the pop-up menu.

Entry delay group

The detection alarm can be included in an entry delay group. Entry delay is a specified time period between the occurrence of an alarm and the annunciation of the alarm. Any alarms in this group are not annunciated immediately upon detection. Instead, the annunciation is delayed by the delay time defined for the group. At the end of the delay time the alarm is annunciated, even if the alarm condition has already been reset during the time period. The alarm will not be annunciated if it has been accessed during the entry delay time period.

Select the **Entry Delay Group** function and enter a group number. If you don't want the alarm to be in a group, specify 0.

Exit delay group

The detection alarm can be included in an exit delay group. Exit delay is a specified time period between triggering the entry/exit control point and the securing of the alarm. If an alarm occurs and is reset within this time, the alarm is not annunciated. Select the **Exit Delay Group** function and enter a group number. If you don't want the alarm to be in a group, specify 0.

Timed access group

The detection alarm can be included in a timed access group. Timed access is a specified time period that an alarm point remains in the access state. The accessed alarm is automatically placed in the secure state at the end of the time period. Select the **Timed Access Group** function and enter a group number. If you don't want the alarm to be in a group, specify 0.

Secondary alarm group

The sensor alarm can be included in a secondary alarm group. If the alarm is not processed (i.e., accessed, secured or released) within a specified time, it will be annunciated at a remote location. Select the **Secondary Alarm Group** function and enter a group number. Enter 0 if you don't want the alarm to be included in a group.

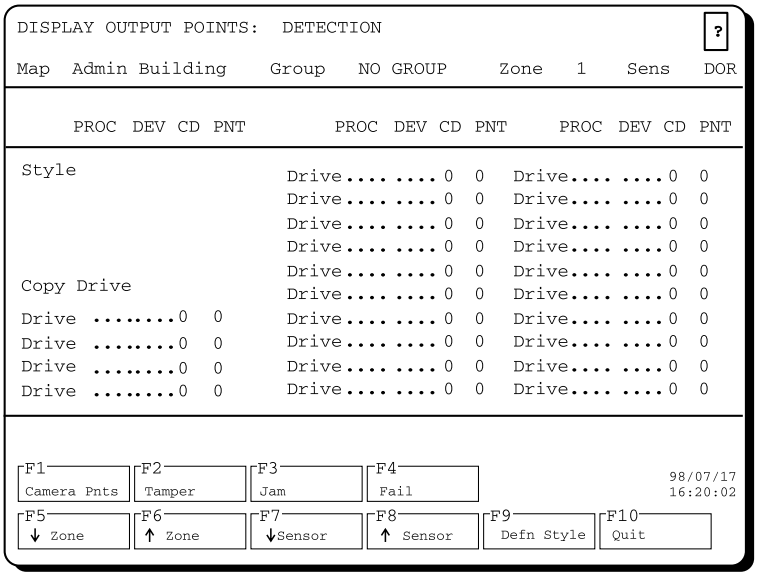
Assigning output points

An output point is a physical point on a hardware device that responds to data coming in from the associated alarm. It's activated based on the state that the display alarm is in. For example, if a display alarm is in an alarm state, the output points associated with the alarm state could be activated to light up a mimic board.

For display alarms, output points can be activated on the following states:

State	Description
Alarm •	the display alarm changes from a secure state to an unacknowledged state
Unacknowledge •	the display alarm changes from a secure state to an unacknowledge state
Acknowledge •	the display alarm changes from an unacknowledge state to an acknowledge state
Access •	the display alarm changes from any state to an access state
Secure •	the display alarm changes from any state to a secure state
Current •	the state where the point is being viewed by the operator (i.e., in alarm processing, acc/sec and video stepping)
Entry delay •	the alarm changes from a secure state to an entry delay state
Exit delay •	the alarm changes from an access or timed access state to an exit delay state
Timed access •	the alarm changes from any state to a timed access state
Input •	this output point follows the state of the alarm input point
Selected •	the display alarm is selected at a Sennet Control Panel

To enter output points, press **F1 - Output Pnts** from the Control Points menu. The screen displays the Output Points menu.



In this menu you can:

- enter the output points on the menu
- define a style form for the menu
- select a style that you've already defined for the menu

Style forms are practical if you use the same point information for alarms of the same type. For example, if you use the same point information for all sub-types of display alarms — detection, tamper, jam and fail — you can create a style form and use it for all the alarms. This saves you from having to enter point information for each sub-type.

You cannot use the same style form among alarm types - i.e., you can't use the same style form for display alarms and remote alarms.

Entering output points on the menu

You can create up to eight styles for each alarm type. To enter the output points:

1. Select the first **Drive** field and pick a state from the pop-up menu Ð alarm, unacknowledge, acknowledge, secure, access, current, input, entry delay, exit delay, timed access, and select. The selected state appears in the **Proc** field. Input fields appear for the device name, card and point number, and the cursor is automatically moved to the **Dev** field.
2. Enter the data in the input fields.

Repeat this procedure for each output point, or use the Copy Drive function to copy the data from one output point to another.

Copying output points

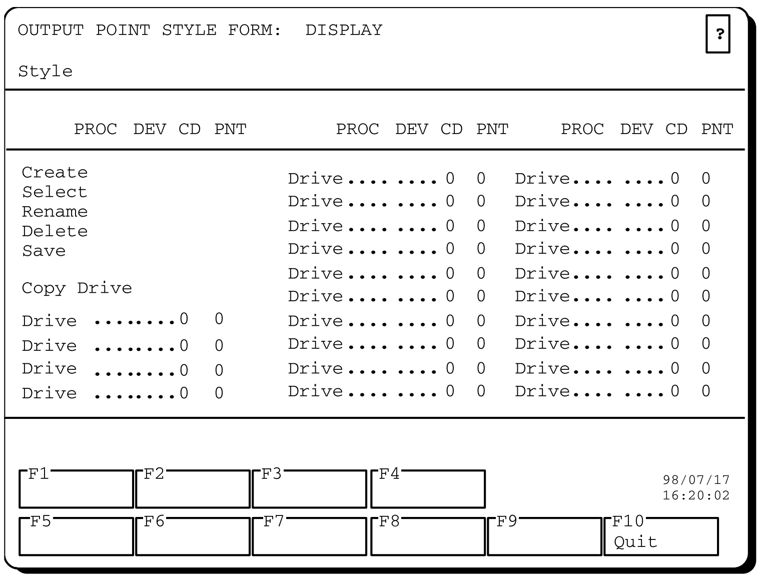
To copy output points:

1. Select the **Copy Drive** function. You're prompted to pick the reference drive.
2. Select the output point that you want to copy from. A flashing * appears beside this point. You're prompted to pick the destination drive.
3. Move to a blank output point and press the **pick** key. The data from the reference point is copied here. The point number increases by 1.
4. Repeat step 3 for as many outputs as you want. Each time you move to a blank output and press **pick**, the point number increases by 1.
5. Press the **Enter** key when you've finished copying.

Defining style form

If you use the same point data for more than one alarm, you can set up a style form for the data. With the style form, you enter the data once, then call up the form for each alarm that uses the data. Style forms are accessed through the alarm output menus.

To set up a style form, press **F9**.



The top line displays the type of alarm that you are creating a style for - in this case, display alarms. This style form can be used for all the display alarms, but cannot be used for any other alarm type.

To create a new style form, select the **Create** field and enter a style name (maximum eight characters) in the input field.

Enter the output point data for each output point, or enter the data for one point, then use the Copy Drive function as described earlier.

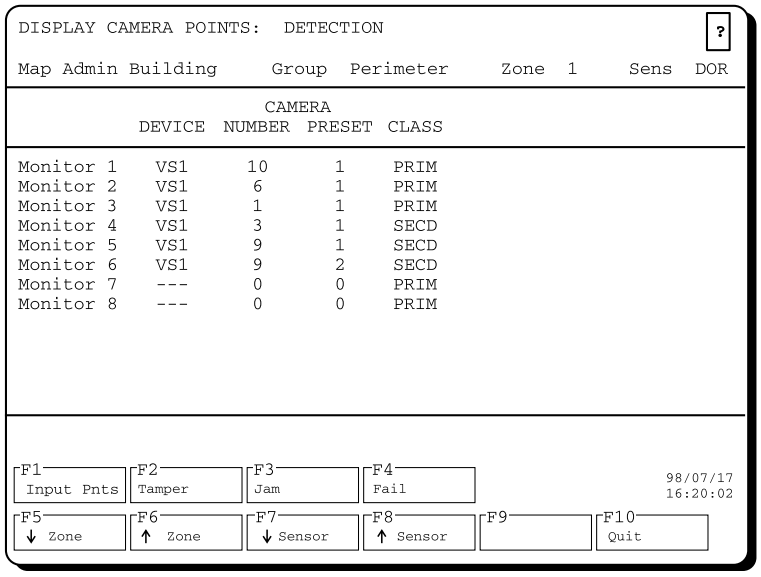
Ensure that you save the style form when done. You can modify it at any time.

Using a syle form

To use a style form, return to the output points menu. Select the **Style** function and choose a style from the pop-up menu.

Entering camera points

To enter camera points, press the **F1 - Camera Pnts** function from the Display Output Points menu. The function can only be accessed if a video switcher has been defined as a primary device. The screen displays the Camera Points menu.



Up to eight cameras can be driven by a display point and displayed on eight monitors. For each monitor, specify the camera number, preset (camera view) and camera classification (primary or secondary). The camera number (1-999) and the preset (1-64) are the numbers you assigned when drawing the maps.

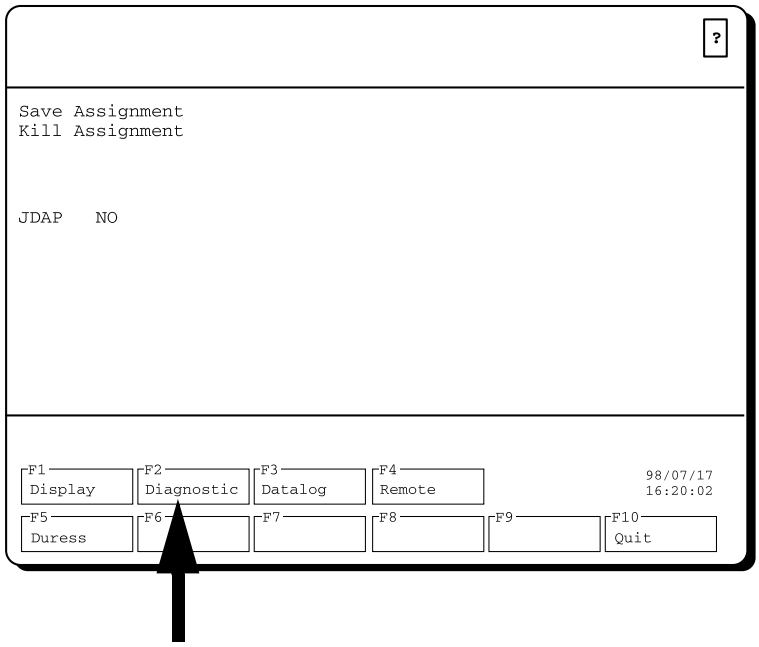
Once you've entered your point and camera assignments for the detection alarms, you can either assign the points and cameras for the tamper, jam and fail alarms, or assign points and cameras for another display alarm.

To assign the points and cameras for the tamper, jam or fail alarms specified for the current display alarm, select **F2** for tamper, **F3** for jam or **F4** for fail. Follow the procedure outlined for assigning detection alarm points.

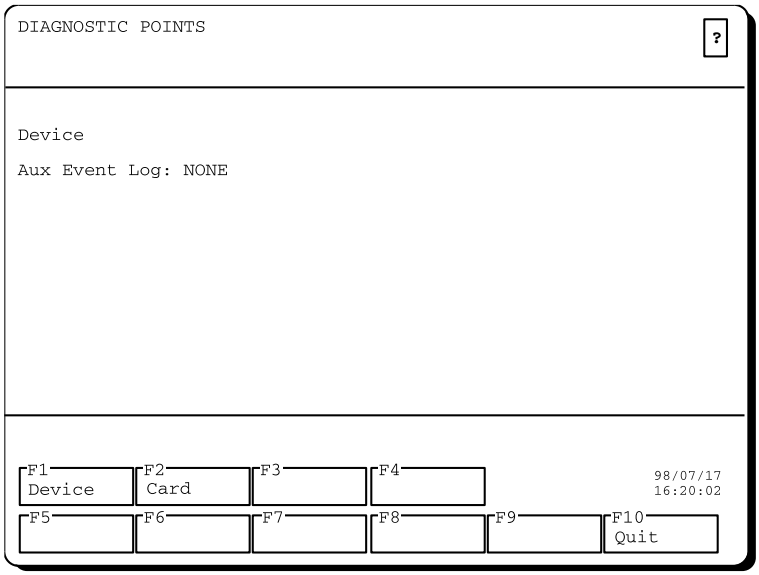
To assign detection alarm points for another display alarm, use the **F5** and **F6** keys to select another zone, and the **F7** and **F8** keys to select another sensor. Enter the points and cameras as explained in the above procedure.

Assigning points for diagnostic alarms

To assign points for the diagnostic alarms, choose **F2 - Diagnostic** from the Assign Points menu.



The screen displays the Diagnostic Points menu.



Selecting a device

To assign points to a diagnostic alarm, you must first select the device. Pick the **Device** function and select a primary device from the pop-up menu.

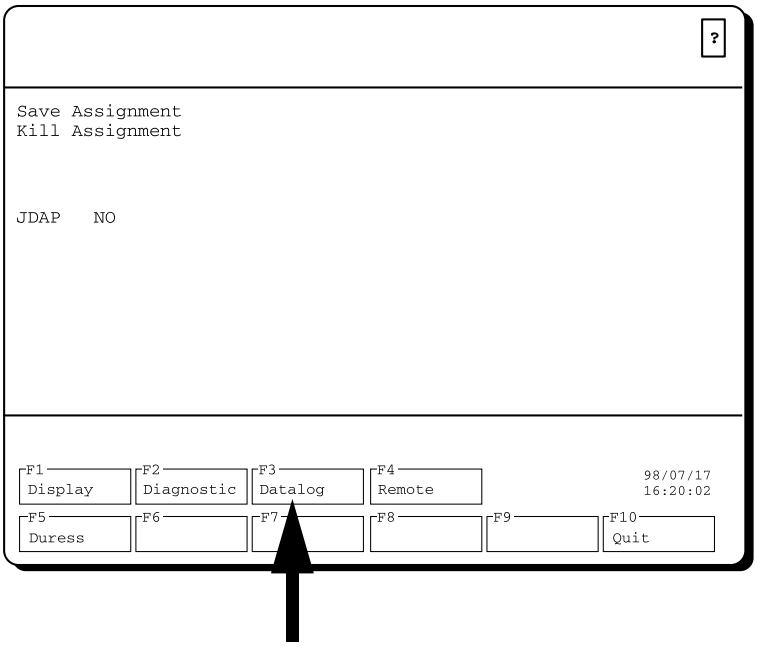
Assigning outputs

Once you've chosen the diagnostic alarm, you must assign output points for the device (and card if the device is a CM or SN). Each device has only one sub-type of alarm - Diagnostic — and the input is automatically defined by the system.

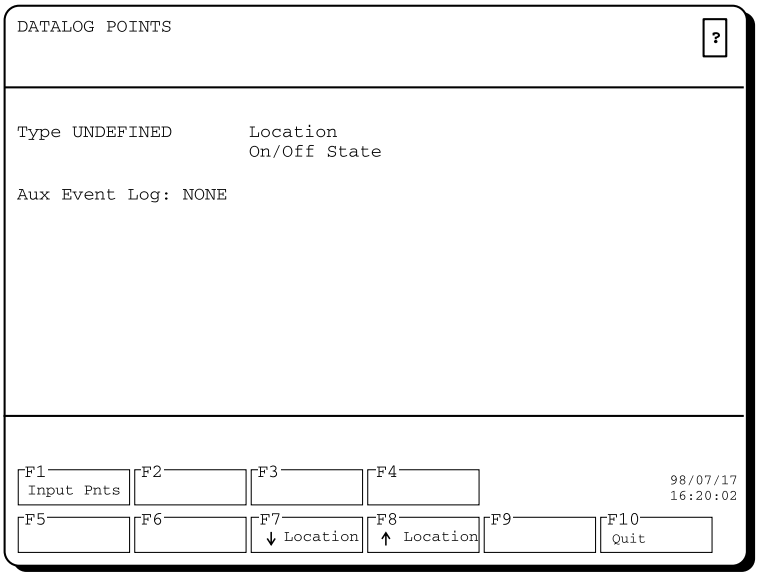
Assign outputs following the same procedure as you did for display alarms.

Assigning points for datalog alarms

To assign points for the datalog alarms, choose **F3 - Datalog** from the Assign Points menu.



The screen displays the Datalog Points menu.



Selecting a datalog alarm

To assign points to a datalog alarm, you must first select the alarm. Choose the **Type** function and select a datalog type from the pop-up menu. The system automatically displays the first alarm data (on/off states and location).

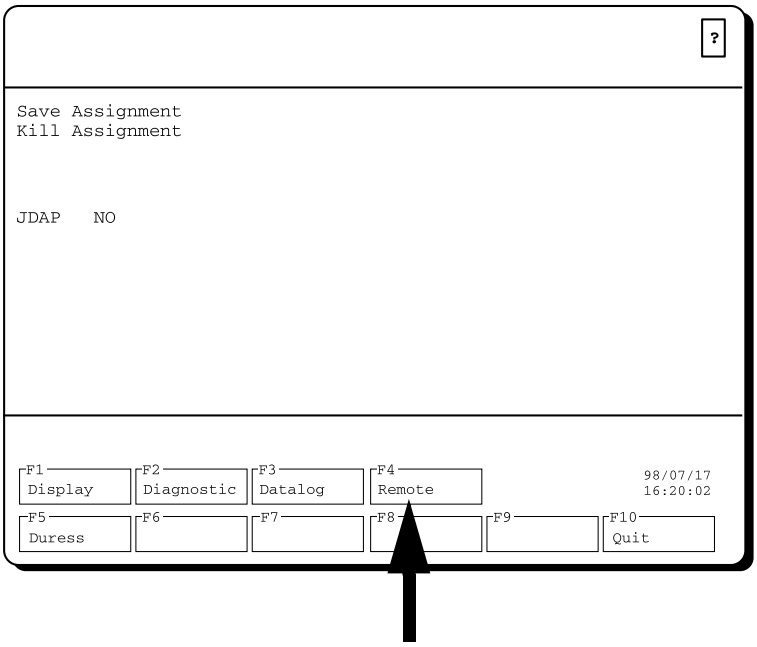
To select another datalog alarm that uses the same type, use the **F7** and **F8** keys to move through the locations. As you move through each location it is displayed in the Location field.

Assigning inputs and outputs

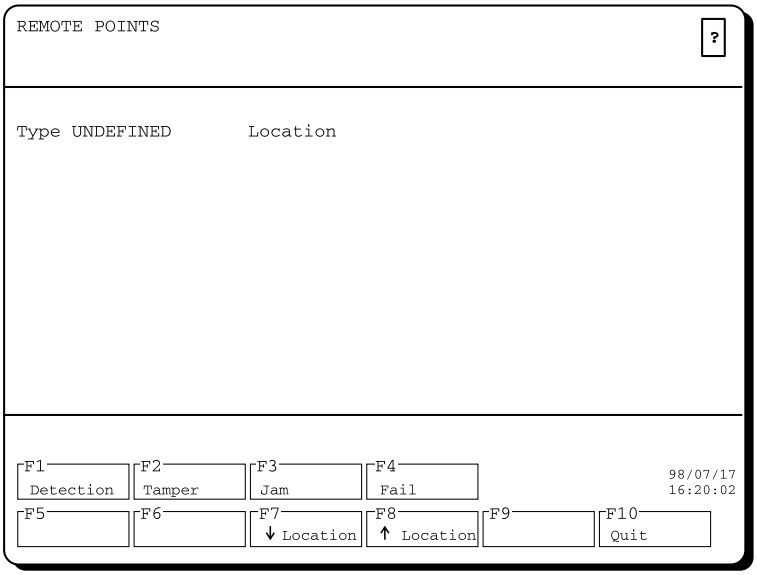
Once you've chosen the datalog alarm, you must assign input and output points for it. Follow the same procedure as you did for display alarms.

Assigning points for remote alarms

To assign points for your remote alarms, choose **F4 - Remote** from the Assign Points menu.



The screen displays the Remote Points menu.



Selecting a remote alarm

To assign points to a remote alarm, you must first select the alarm. Choose the **Type** function and select a remote type from the pop-up menu.The system automatically chooses the first location and displays it in the Location field.

To select another remote alarm that uses the same type, use the **F7** and **F8** keys to move through all the locations connected with that alarm. As you move through each location it is displayed in the Location field.

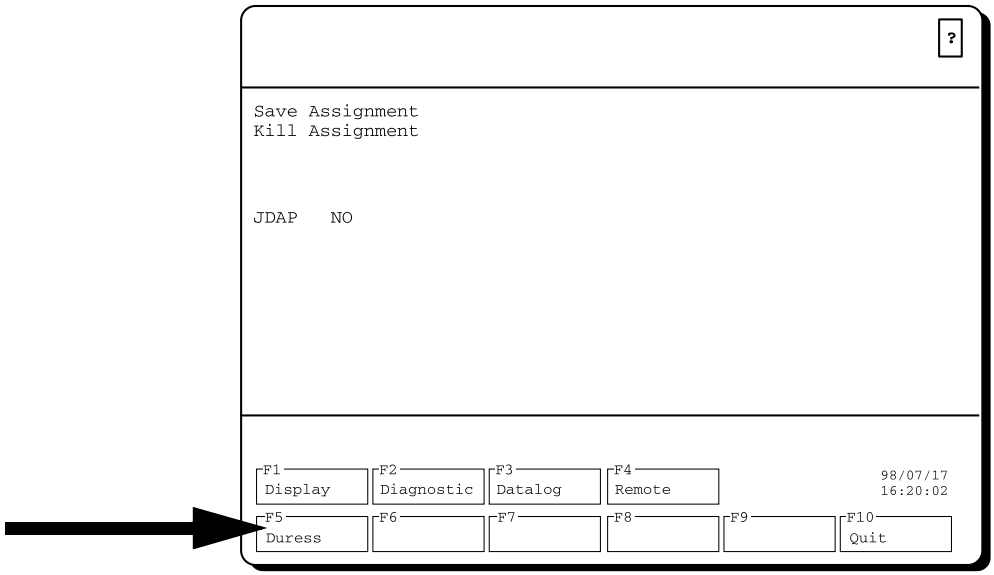
Assigning inputs and outputs

Once you've chosen the remote alarm, you must assign points for each sub-type of alarm generated by the remote alarm.

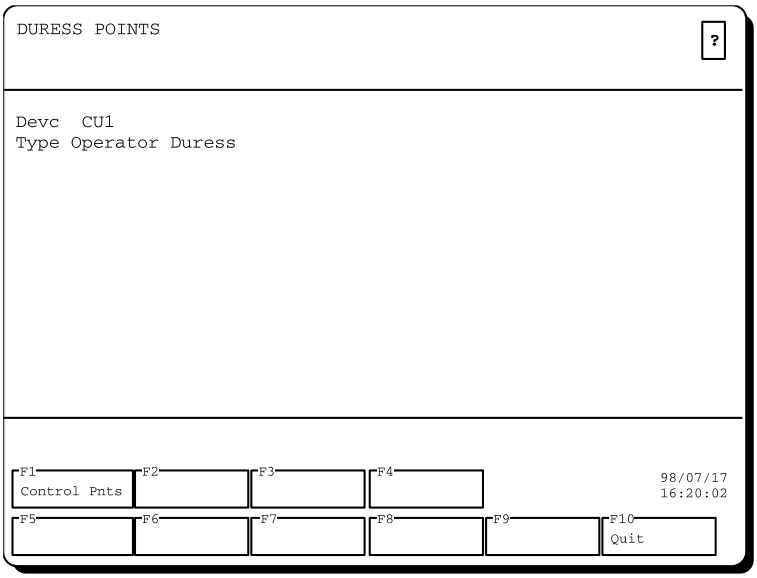
Assign input and output points following the same procedure as you did for display alarms.

Assigning points for duress alarms

To assign points for your duress alarms, choose **F5 - Duress** from the Assign Points menu.



The screen displays the Duress Points menu.



Selecting a device

To assign points to a duress alarm, you must first select the device that is being monitored. Choose the **Devc** function and select either CU1 (the main control unit) or CU2 (the control unit for the redundant system).

Selecting a duress type

Duress alarms are defined by the system. When you enter this menu, the Operator Duress alarm is displayed in the Type field. To choose another duress alarm, pick the **Type** function and select another alarm from the pop-up menu.

Assigning control points and outputs

Once you've chosen the duress alarm, you must assign control points and output points for it.

Assign points following the same procedure as you did for display alarms.

Saving point assignments

To save point assignments for any of the five types of alarms, return to the Assign Points and select the **Save Assignment** function. Do not interrupt the saving process once it has begun. A message will appear once the save is complete.

Deleting all point assignments

To delete all the points you've created, select the **Kill Assignment** function and pick **Yes** from the pop-up menu.

CAUTION	Once you delete the point assignments you cannot recover them. This function deletes all the point assignments even if you've previously saved them with the Save Assignment command.

Editing point assignments

At any time you can enter the Assign Points menu and change the point assignments. You might want to do this if:

- you've modified the configuration of any of the devices
- you've forgotten to enter point assignments for an alarm

To edit the point assignments for an alarm, enter the appropriate menu and key in the new data. To replace existing data, key in the new point information, overwriting the previous data.

Save any changes with the **Save Assignment** command.

7 Translating the Site Creation text

Overview

Once you've created your site, you can translate the site text into another language by using the Language function. This function lets you change the site text into French, German, Italian, Spanish or Arabic, or create a variation of the existing English text by changing the wording.

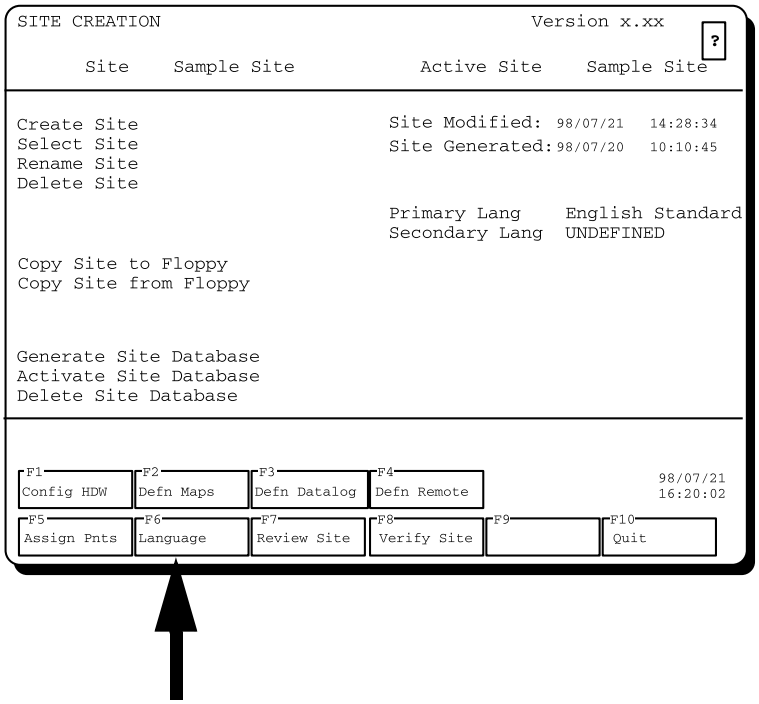
The new language can be used in any of the sites.

The language function lets you translate the text that appears:

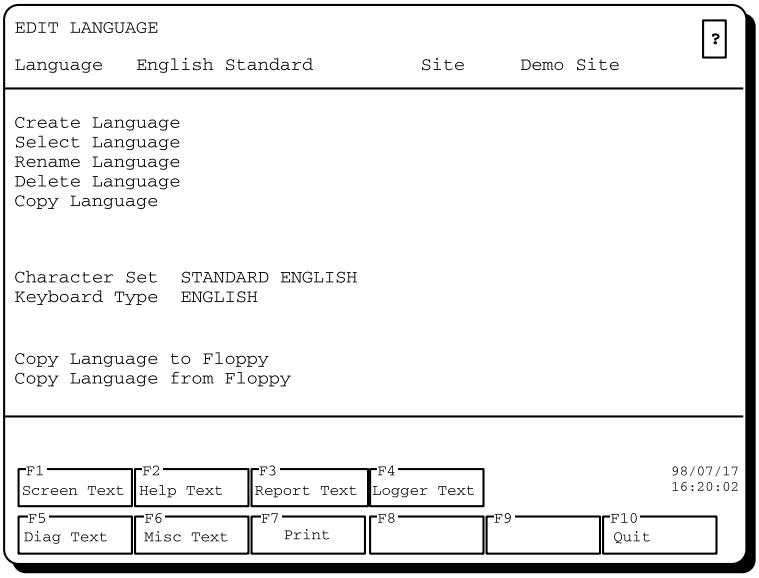
- on the menu screens
- on the help screens
- in the reports
- on the event logger
- during diagnostics

Entering the Language menu

To change the language of a site, select **F6 - Language**, from the Site Creation main menu.



The screen displays the Edit Language menu.



The **Language** field displays the default language (English Standard). You can examine the English Standard text but you cannot modify it.

The **Character Set** field displays the default character set (Standard English), and the **Keyboard Type** field displays the default keyboard type (English).

Creating a new language

To create a new language for the site, you must first name the language. You can call the language anything you choose. The new language can be an actual language (e.g., French), or it can be a variation of the existing text (i.e., the language is still English, but the wording of the text is different).

The **Create Language** function is highlighted when you enter the menu. To specify a new language, press the **pick** key to select the function. Enter the name in the input field (maximum 16 characters).

The new language name appears in the **Language** field, in the current-selection area of the menu.

Loading existing files

When you create a new language, you have no language files to work with. In order to have files to translate, you must first load an existing language to the new language.

To copy existing files to the new language:

1. Ensure that the current language is the one you want to copy to. If not, choose the **Select Language** function and pick another language, or choose the **Create Language** function and key in a new name.
2. Choose the **Copy Language** function and select the language that you want to copy. The selected language copies to the current language.

> If the current language has already been translated, the files will be replaced with the copied files.

Loading files from floppy disk

Language files can also be loaded from floppy disk. Whenever you create or modify a language, you should make a backup on floppy disks. To back up the language files:

1. Select the **Copy Language to Floppy** function on the Edit Language menu.
2. Insert a floppy disk and pick **Backup** from the pop-up menu. The language files are copied to the floppy disk.

You must use a separate floppy disk for each language that you want to back up.

The backup procedure formats the floppy disks; therefore, any data on the disks will be erased.

To reload the files from floppy disk:

1. Select the **Copy Language from Floppy** function.

CAUTION	Make sure that you have selected the appropriate language by choosing it from the Select Language pop-up menu, before choosing this function. This language is the one that you'll load your language files into.

2. Insert the backup floppy disk and pick **Load**. A message will appear when the files are loaded.

Selecting a keyboard character set

Once you've named the language, you must select what keyboard character set you want to use with it. Senstar 100 currently supports three character sets:

- Standard English - is the standard English character set as shown on the English keyboard
- Extended Standard - is the standard English character set plus special characters used in other languages (accents, etc.). These special characters are shown in the adjacent table.
- Standard Arabic - is the standard English character set plus the Arabic character set

> In order to translate to Arabic text, the secondary monitor must be VGA.

Select the **Character Set** function and pick the keyboard character set from the pop-up menu.

Selecting a keyboard type

You must also choose a keyboard type (English, French, German, Italian, Spanish, Arabic).

The keyboard type controls the layout of the characters on the keyboard. The layouts are often referred to by the order of the top 6 characters for example, English - QWERTY, French - AZERTY, German - QWERTZ, Italian and Spanish QWERTY with accented characters.

Select the **Keyboard Type** function and pick the keyboard you want from the pop-up menu.

> The new keyboard character set isn't activated until you generate the site database. See chapter 9, Generating a site database, for information.

Using special keys

If you have the Standard English keyboard, but wish to use special characters such as accents (for example, you want to enter French text using the Standard English keyboard), use the **Alt** key and the specified character codes.

Refer to the table below for a list of special characters.

Code	Character	Code	Character	Code	Character	Code	Character
Alt C ,	Ç	Alt E ´	É	Alt N ~	Ñ	Alt g d	δ
Alt u "	ü	Alt A e	æ	Alt a _	ª	Alt o /	∅
Alt e ´	é	Alt A E	Æ	Alt o _	º	Alt F1	⌈
Alt a ^	â	Alt o ^	ô	Alt Shift?	?	Alt F2	⌉
Alt a "	ä	Alt o "	ö	Alt o o	∞	Alt F3	⌊
Alt a ’	à	Alt o ‘	Ò	Alt g a	∝	Alt F4	⌋
Alt a o	å	Alt u ^	û	Alt g b	ß	Alt F5	⌡
Alt c ,	ç	Alt u ‘	Ù	Alt g g	Γ	Alt F6	—
Alt e ^	ê	Alt y "	ÿ	Alt p i	π	Alt F7	+
Alt e "	ë	Alt O "	Ö	Alt g e	Σ	Alt F9	├
Alt e ’	è	Alt U "	Ü	Alt g s	σ	Alt F10	┤
Alt i "	Ï	Alt a ´	á	Alt g u	µ	Alt F11	⊤
Alt i ^	î	Alt i ´	Í	Alt g t	τ	Alt F12	⊥
Alt i ‘	Ì	Alt o ´	Ó	Alt o |	ϕ		
Alt A "	Ä	Alt u ´	Ú	Alt o -	θ		
Alt A o	Å	Alt n ~	ñ	Alt g w	Ω		

To enter a special character, press and release the **Alt** key, then press the corresponding character keys.

Text that can be translated

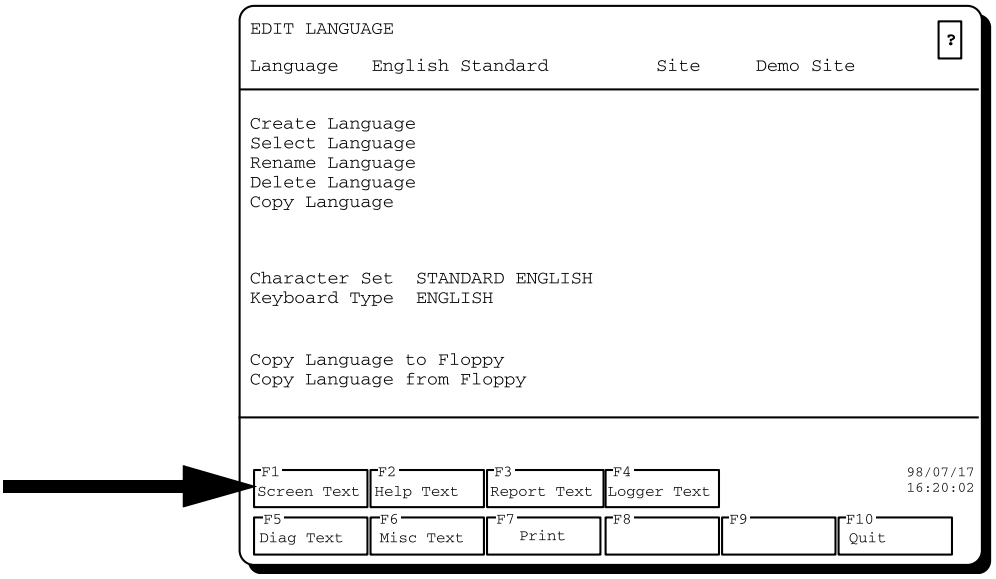
Once you've created the language, select the type of text you want to translate. The function keys display the types of text you can translate or modify:

- screen text (F1)
- help screen text (F2)
- report text (F3)
- event logger text (F4)
- diagnostic text (F5)
- miscellaneous text (F6)

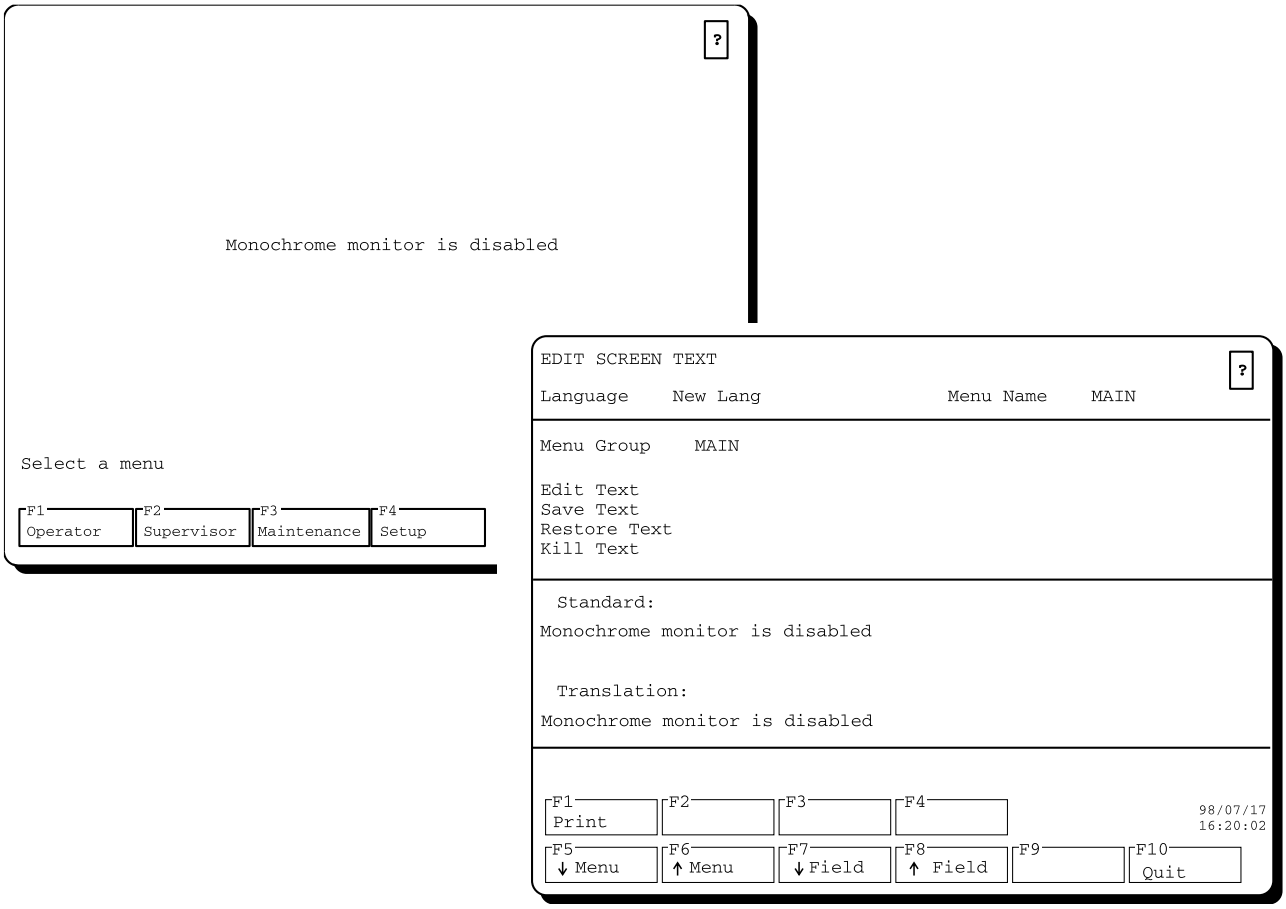
Translating screen text

This section explains how to change the language that appears on the menu screens.

Select **F1 - Screen Text**, from the Edit Language menu.



The first menu appears on the primary and secondary screens.



Selecting a menu screen

To select a menu screen:

1. Choose the **Menu Group** function. A pop-up menu appears, listing the six groups of menus — Main, Alarm, Operator, Supervisor, Maintenance and Setup.
2. Select a menu group from the list. The group name appears in the Menu Group field. The first menu in the group appears in the **Menu Name** field and the menu is displayed on the color-graphics screen.
3. Use **F5** and **F6** to select another menu in the group.
4. Use **F7** and **F8** to choose a field on the menu. The selected field is highlighted on the color screen, and appears on the monochrome screen in the area labeled **Standard** and **Translation**. The Standard field displays the original text. The Translation field displays the translated text. When you first select this menu, the two text strings are the same.

Translating a menu screen

Once you've selected a menu and field:

1. Select the **Edit Text** function. The cursor moves into the Translation field and is positioned at the beginning of the line.
2. Type in the new text, overwriting the text that's already there. You can see the original text in the Standard field and on the color screen. When you reach the maximum number of characters for that text string, the cursor stops moving. The length of the text varies depending on the field you're translating.
3. Press the **Enter** key to complete the function. The revised text appears on the color screen. The text doesn't change in the Standard field.
4. Use **F7** and **F8** to select another field on the menu, or **F5** and **F6** to select another menu. Then pick Edit Text again.

Restoring the original text

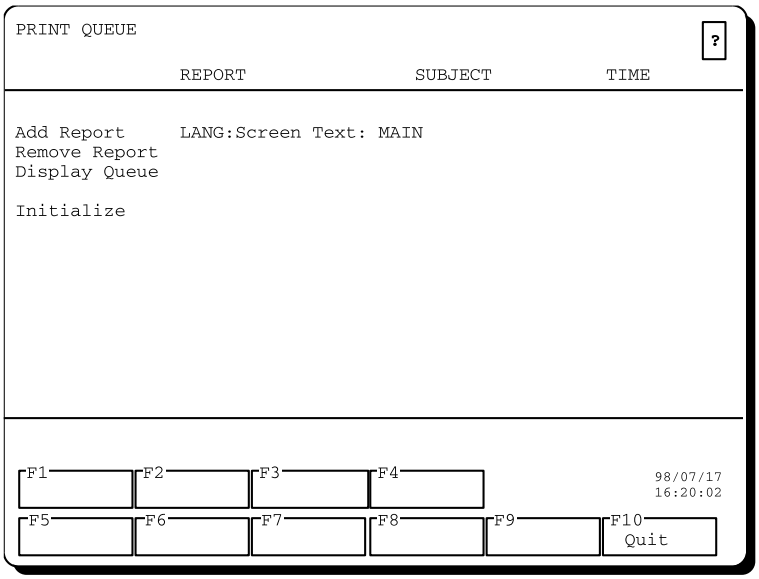
If you change the screen text and want to return to the original version, use the **Restore Text** command. This function restores the text of all menus to the version at the last save.

To restore one field on a menu, you must rekey the field. To restore one menu in a menu group, you must rekey all the fields in the menu.

> Once you've saved the text, you can't use the Restore Text command to restore a previous version. To return the text to a previous version (not the original), you must rekey it. To restore the text to the original version, use the Kill Text function.

Printing screen text

To print all of the screen text for a menu group, press **F1**. The screen displays the print queue menu.



The name of the report you've selected (i.e., LANG: Screen Text:MAIN) appears on the screen.

Select the **Add Report** function. If the Screen Text report is the first in the queue, it will start printing immediately.

Displaying queued reports

To get a display of all reports waiting to be printed, select the **Display Queue** function. A list of all reports in the print queue appears. To exit the Display Queue function, press the **undo** key.

Cancelling the print command

To cancel a print, select the **Remove Report** function. A list of all reports in the print queue appears. Select the report you want to remove and press the **pick** key.

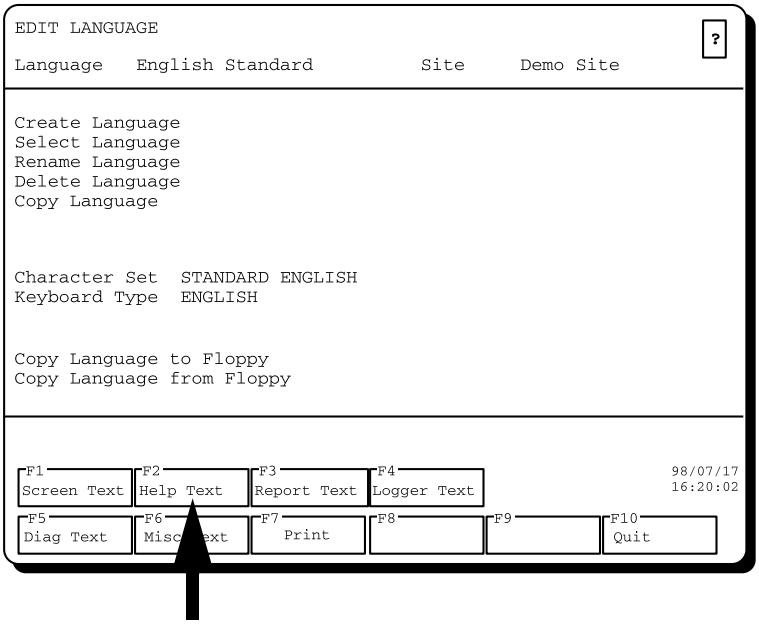
Initializing the printer

To initialize the printer, select the **Initialize** function. The function remains highlighted until initialization is complete.

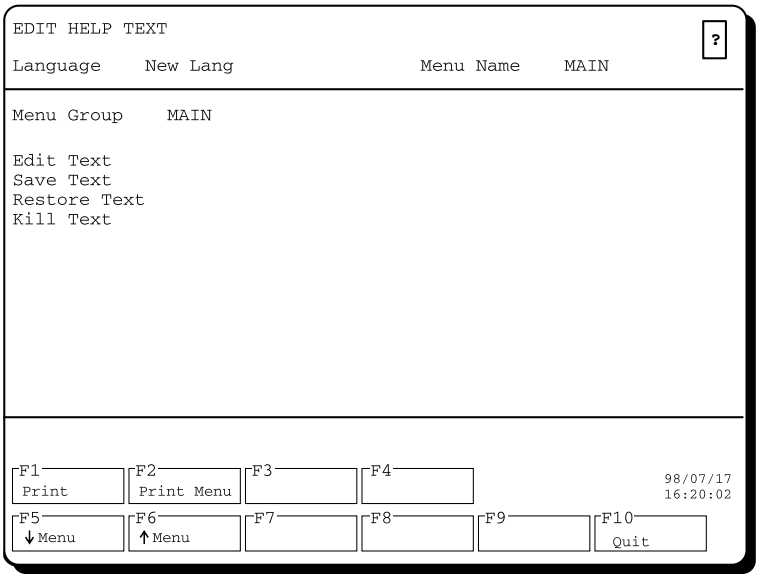
Translating help text

This section explains how to change the language that appears on the help screens.

Select **F2 - Help Text**, from the Edit Language menu.



The screen displays the Help Text menu.

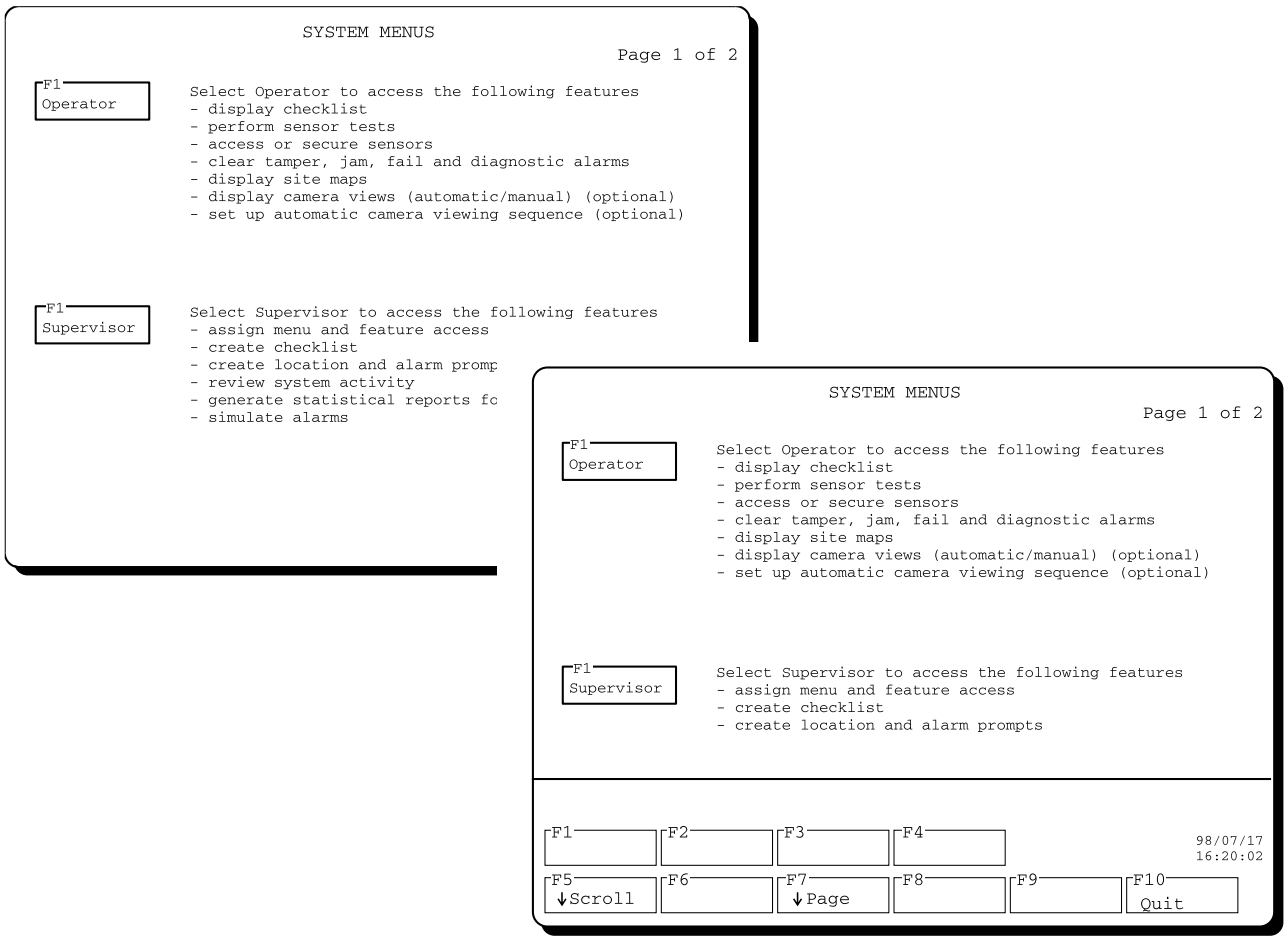


Selecting a help screen to translate

To select a help screen to translate, choose the **Menu Group** function and pick a group from the pop-up menu.

Translating a help screen

Select the **Edit Text** function. The help screen for the menu you've selected appears on the primary and secondary screens.



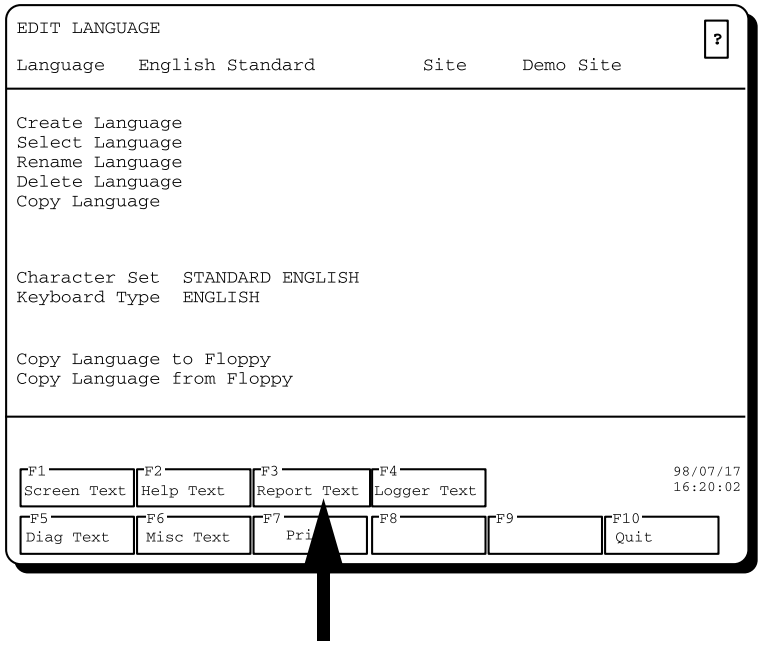
The color screen displays the first page of the menu. The monochrome screen can only display the first 16 lines of the page. The cursor is positioned at the top line of the monochrome screen.

Type in the new text, overwriting the text that's already there. Use the same procedure as described in ***Translating screen text***, page 7-9.

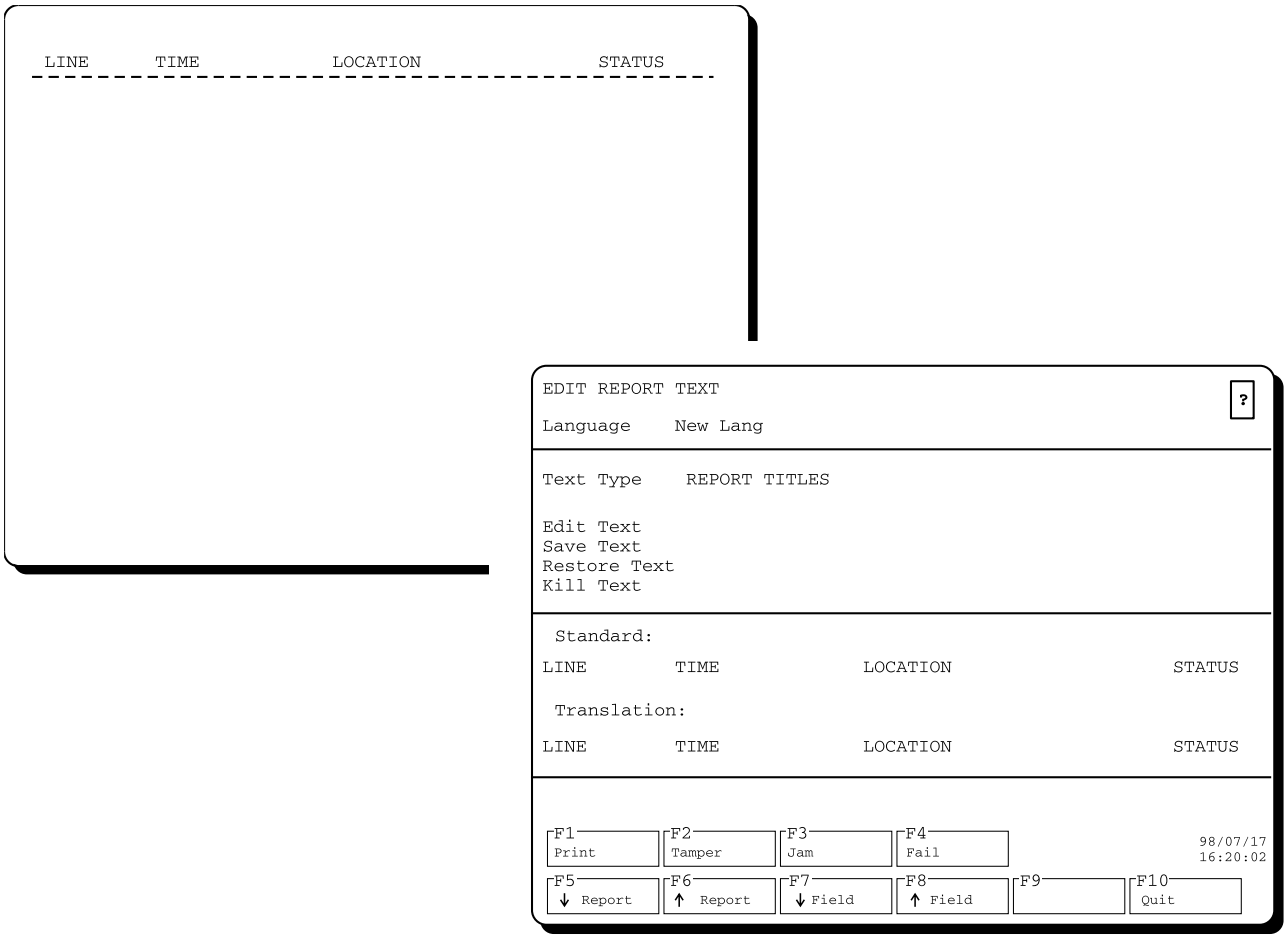
Translating report text

This section explains how to translate the text that appears on the reports. Report text consists of text strings, and when you translate a string, it will be translated in every report that the text appears in. For example, if you change column headings for one report, and those same headings appear in another report, the text will be translated there as well.

Select **F3 - Report Text**, from the Edit Language menu.



The Report Text menu appears on the primary and secondary screens.



Selecting report text

Choose the **Text Type** function. A pop-up menu appears, listing the kinds of report text you can translate. These are:

- Report Titles - report titles and column headings
- Report Text - text that appears in the body of the report
- Report Labels - text that identifies a report when it is sent to the printer

If you choose **Report Titles**, a report is displayed on the color screen. Use the **F5** and **F6** functions to select another report. Use the **F7** and **F8** functions to choose a field in the report. The selected field is highlighted on the color screen, and appears on the monochrome screen in the area labeled **Standard** and **Translation**.

If you choose **Report Text** or **Report Labels**, the text is not displayed on the color screen. The F5 and F6 functions disappear. To choose another field in the Report Text or Report Labels group, use the **F7** and **F8** functions. The selected field appears on the monochrome screen in the area labeled **Standard** and **Translation**.

Translating a report text field

Once you've selected report text to translate, select the **Edit Text** function. The cursor moves into the Translation field and is positioned at the beginning of the line.

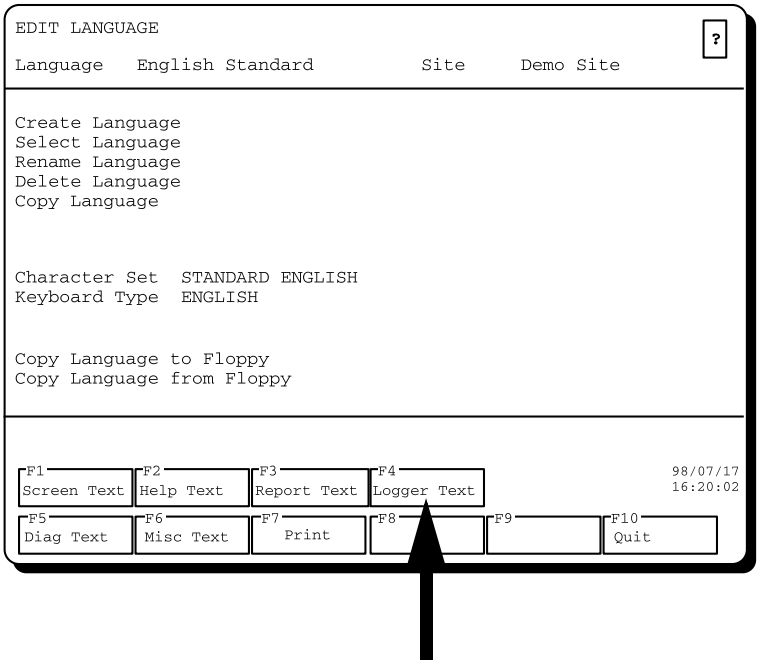
Type in the new text, overwriting the text that's already there.

If you change the length of the column headings, the columns might not line up exactly when the reports are displayed or printed.

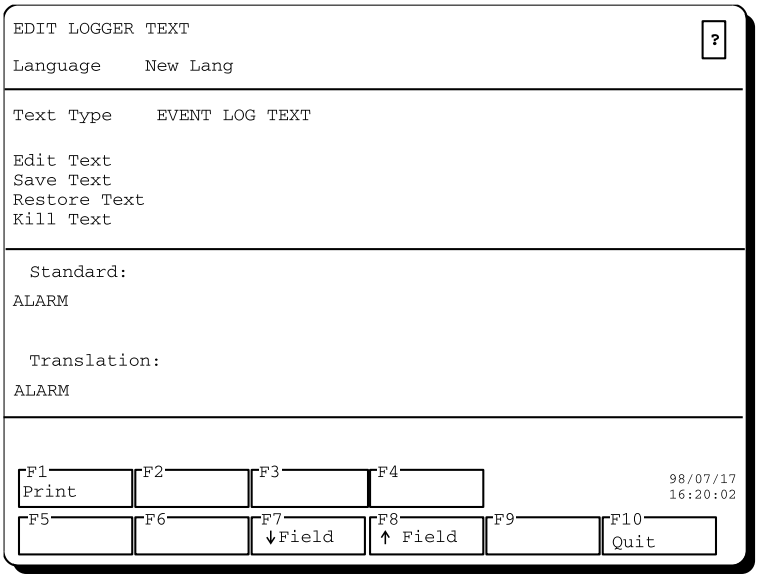
Translating event logger text

This section explains how to translate the text that is printed on the event logger.

Select **F4 - Logger Text**, from the Edit Language menu.



The screen displays the Logger Text menu.



Selecting logger text

Choose the **Text Type** function by pressing the **pick** key. A pop-up menu appears, listing the kinds of logger text you can translate. These are:

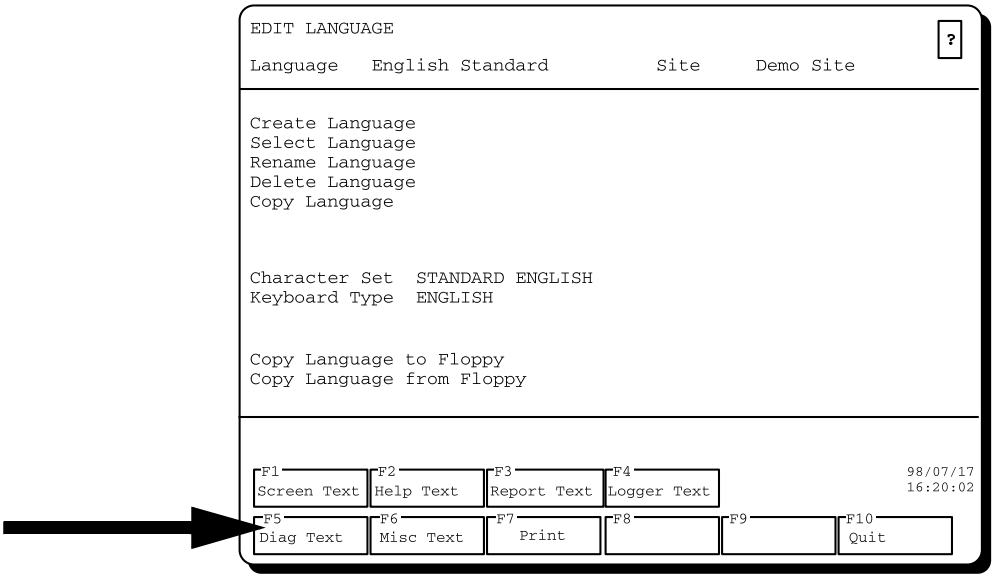
- Event Log Text - portions of text that appear on the event logger under the location column
- Event Log States - status messages that print out on the event logger, under the status column (e.g., logon, logoff)

Select the type of logger text you want to work with. Then select Edit Text to translate it. Use the same procedure as described in ***Translating screen text***, page 7-9.

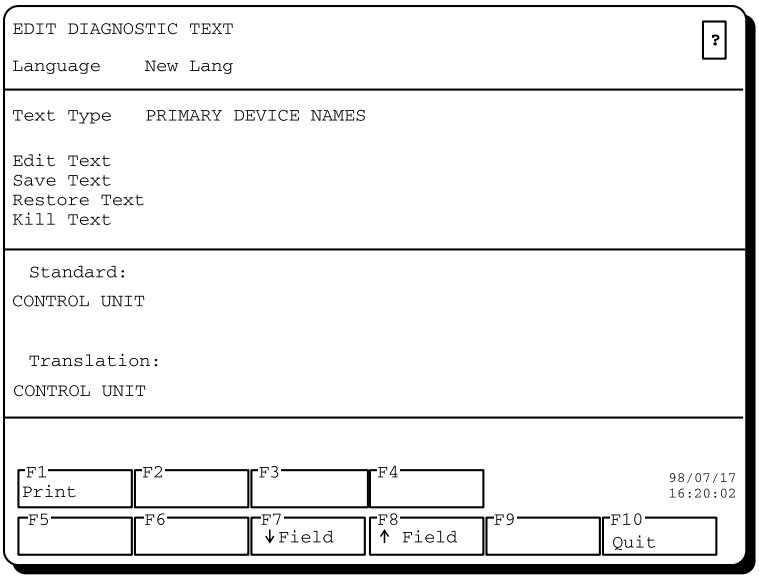
Translating diagnostic text

This section explains how to translate the text that appears during diagnostics. Diagnostic text consists of text strings, and when you translate a string, it will be translated in every place that the text appears in. For example, if you change primary device names, the translated name will appear in every place where the original name appeared.

Select **F5 - Diag Text**, from the Edit Language menu.



The screen displays the Diagnostic Text menu.



Selecting diagnostic text

To select diagnostic text, choose the **Text Type** function. A pop-up menu appears, listing the kinds of diagnostic text you can translate. These are:

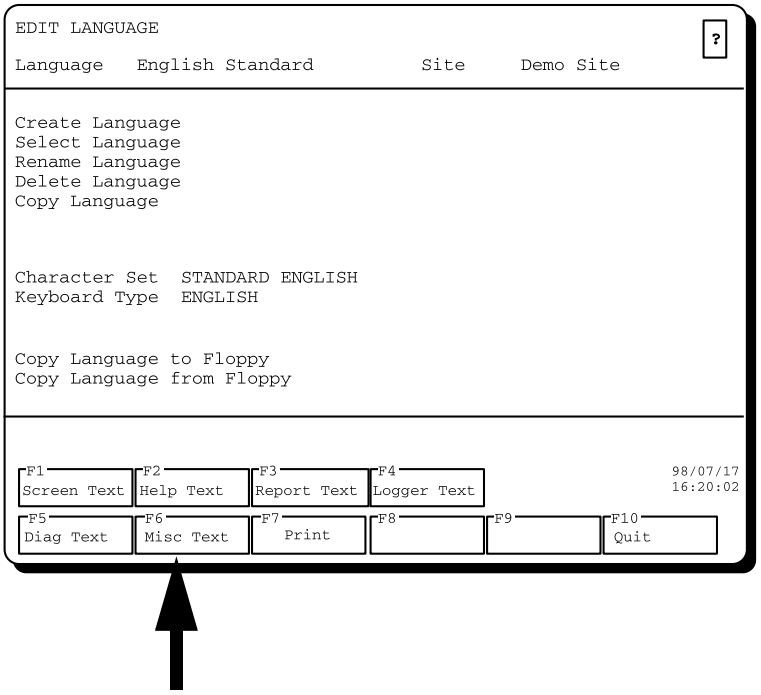
- Primary Device Names - names of the primary devices in your system (e.g., Control Unit 1)
- Primary Device Labels - labels of the primary devices (e.g., CU1)
- Primary Card Labels - labels of the cards for the primary devices CM and SN
- Secondary Device Names - names of the secondary devices in your system (e.g., Event Logger)
- Diagnostic Locations - identifying messages (e.g., Card1)
- Diagnostic States - status messages (e.g., Fail, Fault, Off)

Select the type of diagnostic text you want to work with. Then select **Edit Text** to translate it. Use the same procedure as described in ***Translating screen text***, page 7-9.

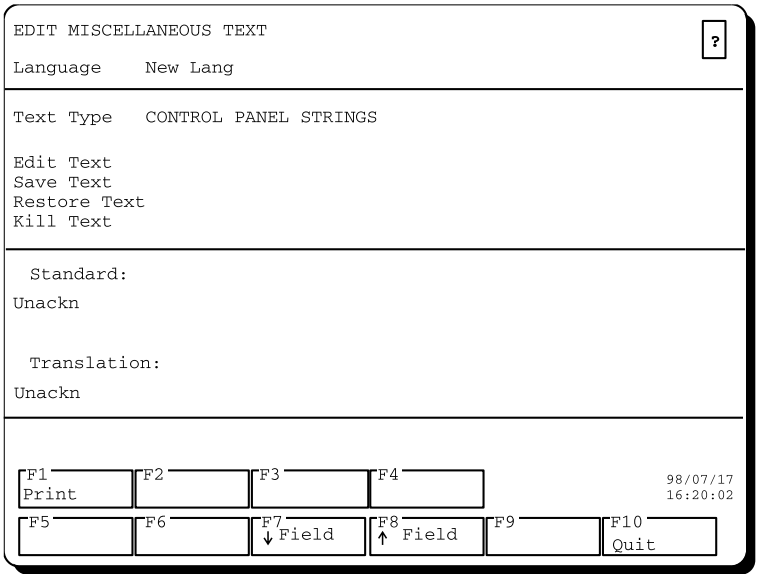
Translating miscellaneous text

This section explains how to translate the miscellaneous text such as the text strings that appear on the control panel.

Select **F6 - Misc Text**, from the Edit Language menu.



The screen displays the Miscellaneous Text menu.



Selecting miscellaneous text

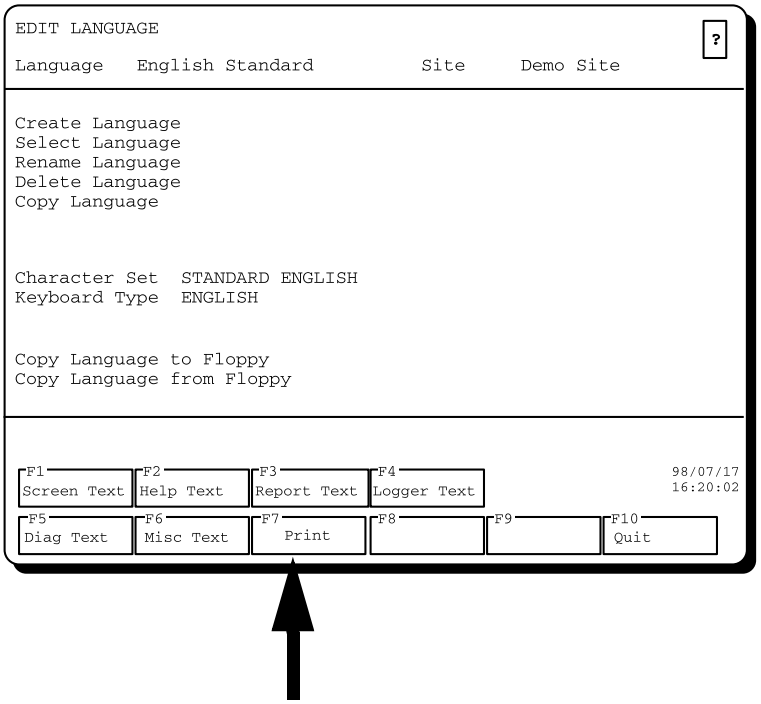
Choose the **Text Type** function by pressing the **pick** key. A pop-up menu appears, listing the kinds of miscellaneous text you can translate. The types of text available for translation will depend on your system.

Select the type of miscellaneous text you want to work with. Then select **Edit Text** to translate it. Use the same procedure as described in ***Translating screen text***, page 7-9.

Printing all language files

You can print all the translated text (screens, help text, reports, event logger text and diagnostic text) by using the Print function in the Edit Language menu.

Select **F6** to print all the text files.



This function puts all the files in the print queue, and prints them one at a time.

> You will need a printer capable of accepting downloadable characters in order to print an Arabic translation.

8 Verifying site data

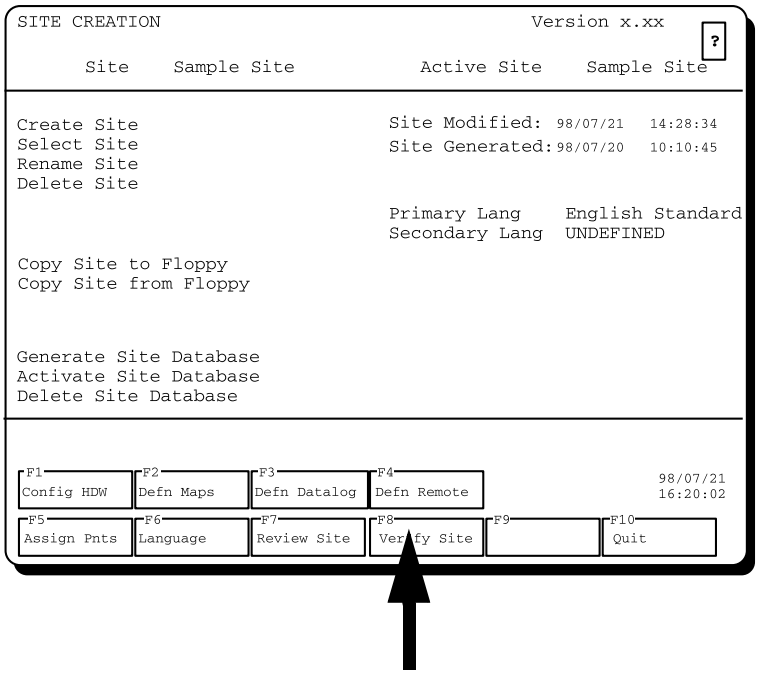
Overview

Once you've entered point assignments for all the alarms, you should verify that the assignments are correct and complete. All points must be assigned properly before your system will work.

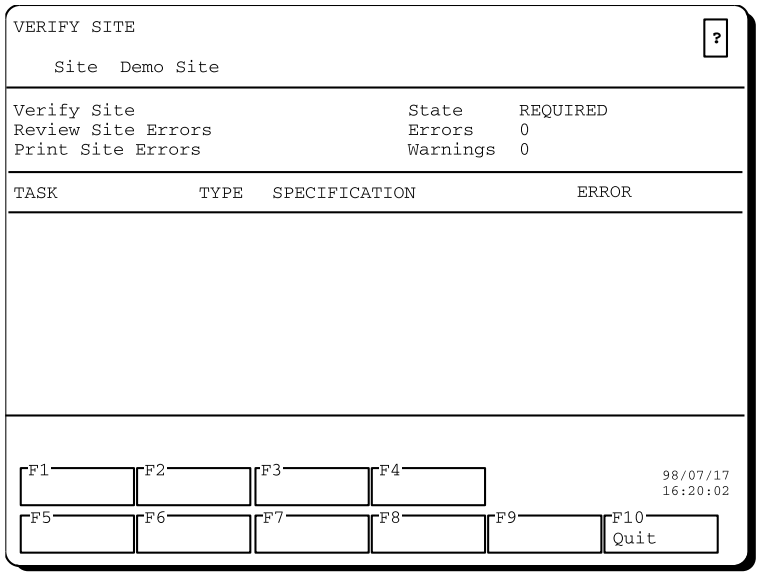
This chapter explains how to use the Verify Site function to check that the site is set up correctly.

Entering the Verify Site menu

To verify the site, select **F8 - Verify Site** from the Site Creation main menu.



The screen displays the Verify Site menu.



The current site is displayed in the Site field. The State field displays **Required**, indicating that you haven't run a verification on the site yet, or that a verification hasn't been done since you made changes to the site.

Verifying the setup of a site

To ensure that the site has been correctly set up, select the **Verify Site** function.

The system automatically runs a check on the site data. When the verification is finished, the message, 'Site Verification Complete!' appears on the prompt line.

The **State** field displays the status of the site — either **Success** if the verification was successful, or **Fail** if any errors were found in the site data.

The **Errors** field shows the number of fatal conditions for the site. These errors will prevent the system from operating. For example, if you have missed placing a primary device symbol on a map, you'll get an error.

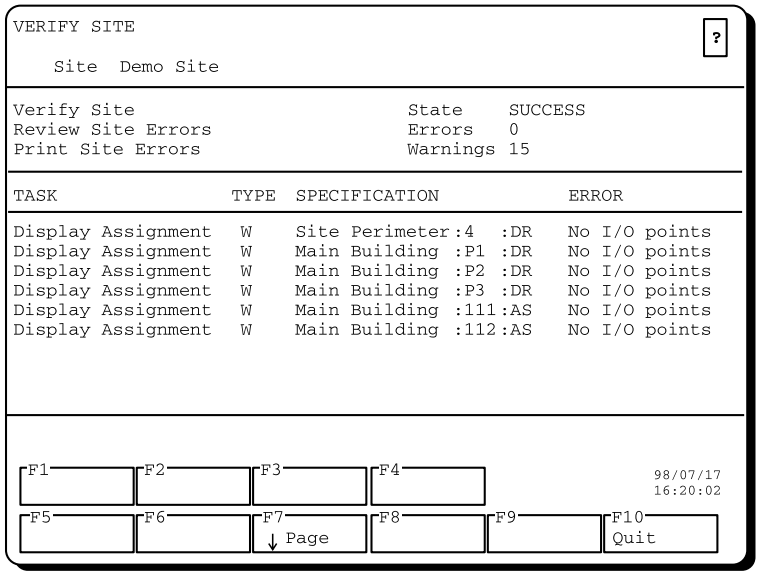
The **Warnings** field shows the number of non-fatal conditions for your site. You might get warnings even if the verification has been successful. The warnings indicate that your system is not fully operational, even though it will run.

Reviewing site errors

If the site verification indicates any errors or warnings, you'll need to examine them and then correct the errors before your system will run properly. You can review the errors or warnings in two ways - you can display them on the screen, or print them out.

Displaying an error report

To display an error report on the screen, highlight the **Review Site Errors** function, and press the **pick** key. The report is displayed on the screen. A sample report is shown below:



The report is set up as follows:

The **Task** column lists the point assignments that have caused the error or warning. The possible assignments are:

- hardware configuration
- maps
- map zones
- display assignment
- datalog points
- datalog assignment
- remote points
- remote assignment

The **Type** column indicates whether the condition is an error or a warning — E for error, W for warning.

The **Specification** column lists the point specifications. If the condition concerns a display alarm, the map, group, zone and sensor name are shown. If the condition concerns a datalog or remote alarm, the type and location data are shown, etc.

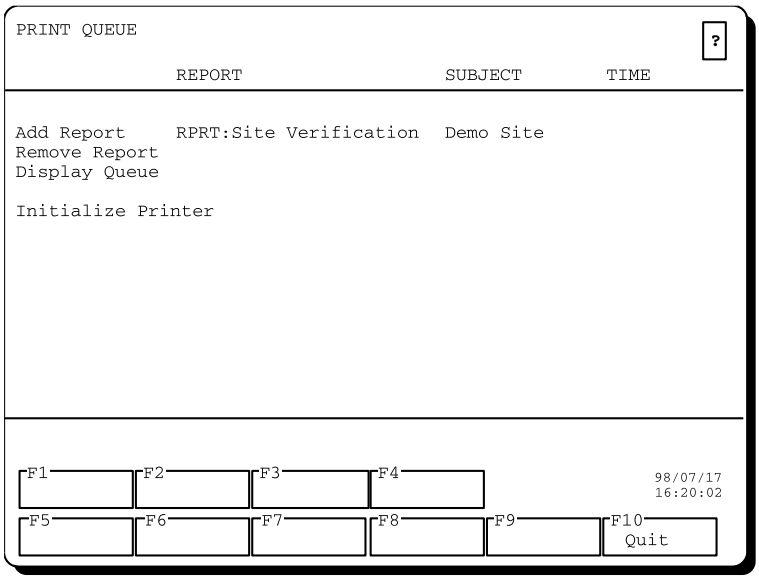
The **Error** column lists the error or warning associated with the alarm. The following table lists of some of the possible errors and warnings for each task.

Task	Errors/Warnings	Symbol
Hardware configuration	• No port	E
	• No memory	E
	• No primary device	W
	• No cards	W
Maps	• Group not used	E
	• No zones	E
	• No maps	E
	• No device icon	E
Map zones	• No assignment	E
Display assignment	• No primary map	E
	• No I/O points	W
Datalog assignment	• No input points	W
Datalog points	• No assignment	W
Remote assignment	• No input points	W
Remote points	• No assignment	W

If the error report is more than a page in length, the **F7** and **F8** page functions appear. Use F7 and F8 to scroll through the pages of the report.

Printing an error report

To print an error report, select the **Print Site Errors** function. The screen displays the print queue menu.



The name of the report you've selected (i.e., Site Verification) appears on the screen, as well as the site that you are verifying.

Select the **Add Report** function. If the Site Verification report is the first in the queue, it will start printing immediately.

To get a listing of all reports waiting to be printed, select the **Display Queue** function.

To stop a report from printing, select the **Remove Report** function.

To initialize the printer, select the **Initialize Printer** function. The function remains highlighted until initialization is complete.

Correcting errors

You must correct any errors in the site before your system will work properly. To correct errors:

1. Exit from this menu and return to the Site Creation main menu.
2. Select the appropriate function (e.g., Assign Points, Define Site Maps).
3. Make any changes, e.g., enter point information, reconfigure hardware, add zones.
4. Return to the Site Creation main menu and select the **Verify Site** function to verify the site again. Follow the above steps until the status of the site is **Success**.

9 Generating a site database

Overview

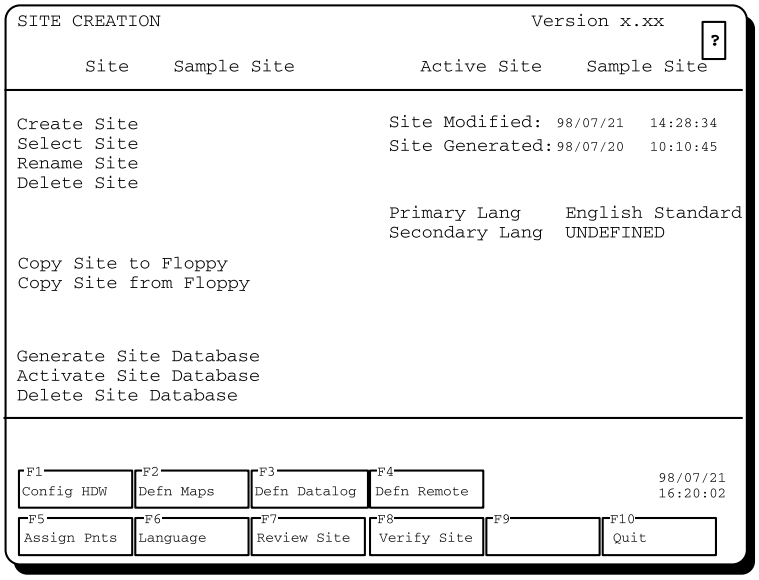
Once you've entered all your site data into the system, and you've verified that the information is correct and complete (refer to Verifying site data, chapter 8), you have to generate a run-time database before the Senstar 100 system can use it.

This chapter explains how to generate a run-time site database of all the data you've entered for the site. It also covers how to activate the database so that it is the one used by the system when you boot up, and how to delete a database. Finally, the chapter contains a reminder of backing up and restoring procedures.

Generating the site database

After you've entered all the site data and have verified it with the Verify Site function, you're ready to generate a database for the site.

To do this, you must return to the Site Creation main menu. Press **F10** to exit from whatever menu you are in, and keep pressing it until you're back at the main menu.



Make sure that the site you want to generate a database for is displayed in the Site field.

Select the **Generate Site Database** function. This function takes the system configuration information and the site maps, and combines everything into one database. This then becomes the site data that will be used by the Senstar 100 application.

CAUTION	This process cannot be halted once it is started. It generally takes from 5 to 15 minutes, depending on the size of the database.

When the database has been generated, the date and time will appear in the **Site Created** status field.

Backing up the site database

After you've generated a site database, you should make a backup of it on floppy disks.

To make a backup of the database:

1. Select the **Copy Site to Floppy** function.
2. Insert a floppy disk and pick **Backup** from the pop-up menu. Depending on the size of your system, you'll need one or more disks for the Site Creation data (the database files), and one or more for the site data (the operating files used by the application program. These are the files created when you generated the database). You'll be prompted to insert each disk in turn. Remember to label each disk and number it properly.

   Backing up the site data files may take up to 30 minutes.

If you haven't yet generated the database, you only need the disks for the Site Creation data.

CAUTION The backup procedure formats the floppy disks; therefore, any data on the disks will be erased.

Activating the site database

In order to have the system boot up the site that's currently displayed in the Site field (i.e., to access the site from the Operator, Supervisor and Maintenance menus), you must activate the site database.

WARNING	You must only activate the site that is specifically designed for your system configuration. If you activate a site that is incompatible with your system hardware, serious system problems might occur. Therefore, if you are creating a database for a target computer, do not activate the site on the computer you're currently working on. Load the database onto the target computer, then activate it from there.

Choose the **Activate Site Database** function.

When you activate a site, the site name appears in the **Active Site** field (current-selection area of the menu).

Wait until the message, "Site Activated" appears on the screen. Then press **F10** to exit from the menu. The system will restart, and will reboot the site you've activated.

Troubleshooting the database

If there are any system unit hardware incompatibilities in the database, the system won't activate the database. To find out what the errors are:

1. Press **F10** to exit from the Site Creation function and return to the system menus.
2. Select the **Maintenance** function (**F3**), then the **View Equipment Configuration** function (**F3**). Any incompatibilities between the hardware in the system unit and the database that you've created, will be identified.

Recovering corrupted data

If the database on the hard disk is ever corrupted (due to a power failure, system failure, etc.), you can restore it from your backup floppy disks.

To restore the database:

1. Select the **Copy Site from Floppy** function.

> **CAUTION** Make sure that you've selected the appropriate site before choosing this function.

2. Insert the first of your Site Creation data disks and pick **Load**. When the data is loaded, you'll be prompted to enter any other disks as required.

This function restores the Site Creation data only. To restore the site data disks, you must load them from the Startup menu, or select the Generate Site Database function and generate a new database.

Deleting a site database

Your system can contain up to eight sites. If you need more space on the hard disk, you can delete unused databases.

To delete the database for the site displayed in the Site field, select the **Delete Site Database** function.

This function deletes ONLY the site data files and NOT the Site Creation data. These site data files can be regenerated by selecting the Generate Site Database function.

Deleting an entire site

Use the Delete Site function to delete **both** the Site Creation data and the site data files.

WARNING	If you use this function, all data for the site will be lost.

10 Viewing site reports

Overview

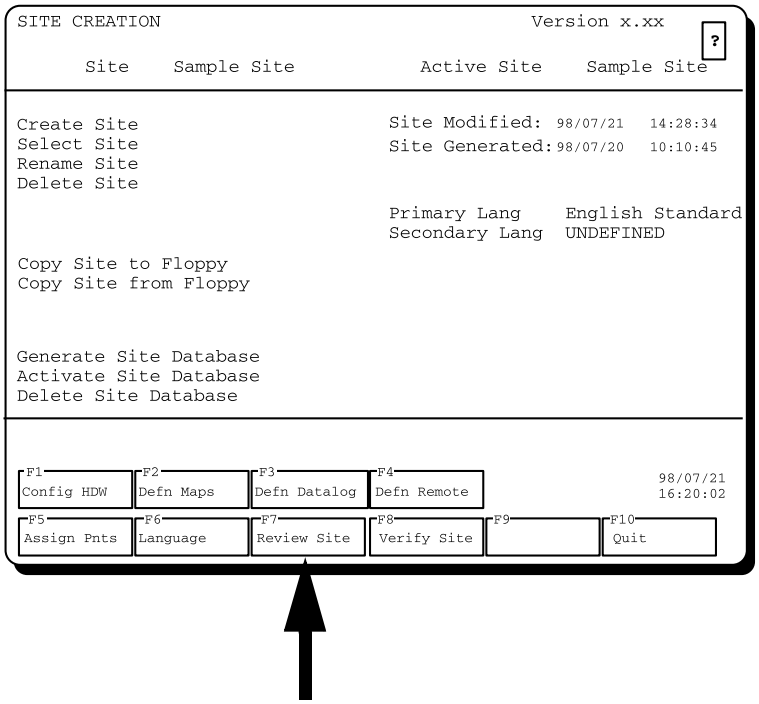
The Senstar 100 Site Creation package includes a report facility which lets you display or print out reports of your system. This function lets you review the hardware configuration and point assignments for all the alarms.

You might find it helpful to view reports of the site before generating a site database, in order to ensure that your system is set up the way you designed it. The report facility is also useful for generating printouts of the point assignments, which you can use to connect all your equipment. However, reviewing reports is not a **required** step in setting up the database.

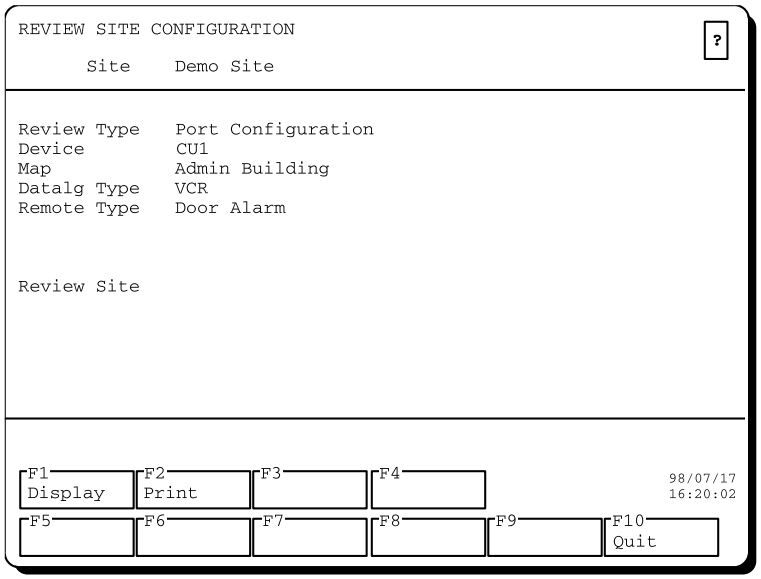
This chapter describes the types of reports that you can generate.

Entering the Review Site menu

To review the site, select **F7 - Review Site**, from the Site Creation main menu.



The screen displays the Review Site Configuration menu.



Selecting a report

There are 17 types of reports that you can view.

Report	Description
Port configuration	lists all the cards in a control unit, the ports being used and the devices attached to them, and the communications parameters for each device
Input configuration	lists the input point configuration for a selected device
Input assignments	lists the input point assignments for a selected device
Display inputs	lists the input point assignments for display alarms (map, group, zone and sensor) on a selected map
Datalog inputs	lists the input point assignments for the datalog alarms
Remote inputs	lists the input point assignments for the remote alarms
Duress inputs	list the input point assignments for the duress alarms
Output configuration	lists the output point configuration for a selected device
Output assignments	lists the output point assignments for a selected device
Display outputs	lists the output point assignments for display alarms on a selected map
Diagnostic outputs	lists the output point assignments for the diagnostic alarms
Datalog outputs	lists the output point assignments for the datalog alarms
Remote outputs	lists the output point assignments for the remote alarms
Duress outputs	lists the output point assignments for the duress alarms
Camera assignments	lists the assignments for each camera
Display cameras	lists the camera assignments for display alarms on a selected map
Diagnostic cameras	lists the camera assignments for diagnostic alarms

Viewing reports

To view any of the reports:

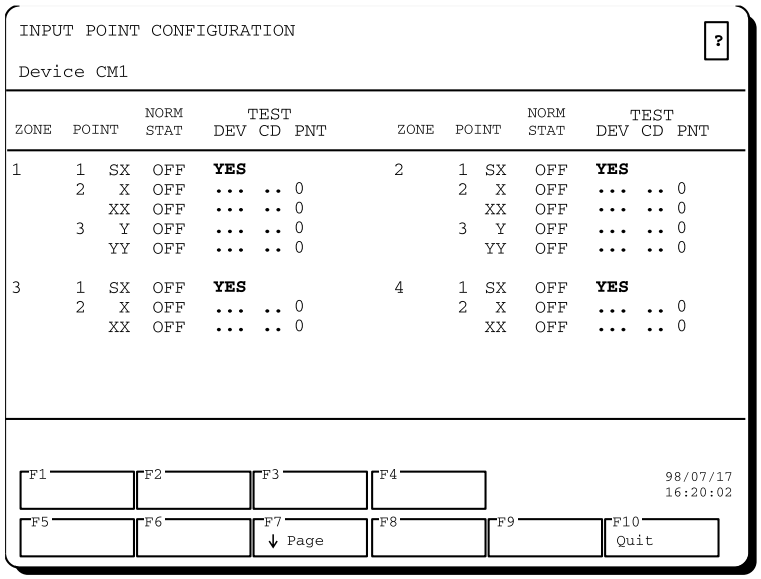
1. Select the **Review Type** function and pick a report from the pop-up menu.
2. Asterisks (*) appear beside the functions that are related to the report, indicating that you need to choose these functions to select more information (i.e., you might need to specify a device name, map name, datalog type or remote type).
3. Once you've picked a report to view, select the **Review Site** function. The system automatically generates the specified report.
4. Choose **F1** to display the report, or **F2** to print it.

A sample report is shown on the following page.

Sample input configuration report

To review the input point configuration for a particular device:

1. Select the **Review Type** function.
2. Select the **Input Configuration** report from the pop-up menu.
3. Select the **Device** function and choose a device from the pop-up menu. You can only display reports for devices that have been used for point assignments. You cannot display reports for CU1, CU2 or VS1.
4. Select the **Review Site** function to generate the report.
5. Choose **F1** to display the report, or **F2** to print it. A sample report follows:



The **Zone** column displays the TM zone number.

The **Point** column displays the point number for the card.

The **Norm Stat** column displays the normal state for the point.

The **Test** columns display the device, card, and point number for the self test point.

a Installing the software

Installation requirements

The Senstar 100 system comes with Senstar 100 application software disks and a sample site data disk which contains a sample site. This software should be installed when you receive your system; if it is not, install it by following the instructions in the Senstar 100 Installation Guide.

The Site Creation software consists of:

- Site Creation disks that contain the operating files needed to run the site creation application
- one Sample Site Creation data disk that contains a sample site database

The Site Creation software should be installed when you receive your Site Creation system. If it is not, perform the following procedure to install it.

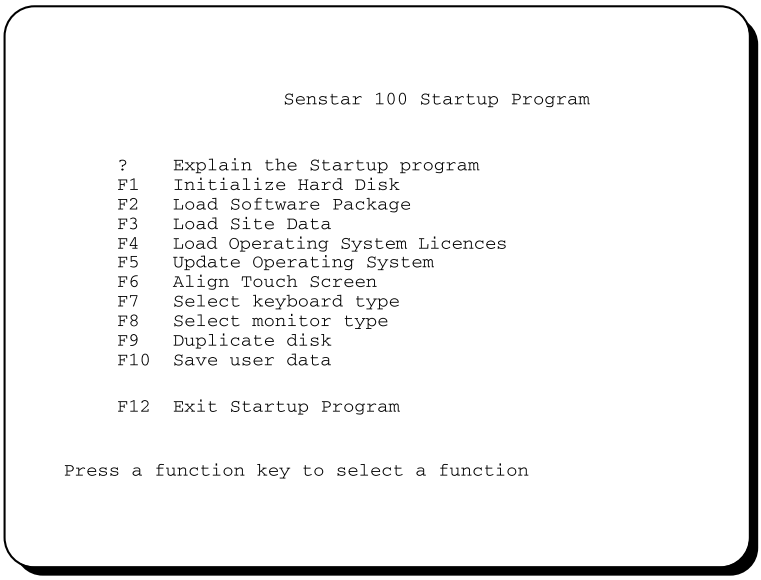
Loading the Site Creation software

Follow this procedure if the Site Creation software is NOT installed on your system.

To load the Site Creation software:

1. Boot the system with the QNX™ boot disk:
   a. Insert the floppy disk labeled **QNX O/S and Utilities Disk 1 of 2** in drive "A".
   b. Turn on power to the computer.

The screen displays the Startup menu.



2. Press the **F2** function key to select **F2 - Load Software Package**.
3. Insert the first disk of the Site Creation software into drive "A". Press the **space bar** when ready to continue.
4. When the disk is loaded, remove it and insert subsequent disks into drive "A". Press the **space bar** when ready to continue. Repeat this procedure until all disks are loaded.
5. When loading is complete, press the **space bar** to return to the Startup menu.
6. Remove the Site Creation software disk from the drive.

Backing up the Site Creation disks

You should make backup copies of the Site Creation software disks and the Sample Site Creation data disk. To make backups:

1. Select **F8 - Duplicate Disk.**
2. Insert the disk to be duplicated into drive "A". Press the **space bar** when ready to continue.
3. Follow the instructions on the screen. When the disk is backed up, remove it and follow the same procedure for the other disks.

Exiting from the Startup menu

1. Press the **F10** function key to exit from the Startup menu.
2. Reboot from the hard disk:
   a. Remove the floppy disk from drive 'A'.
   b. Turn off the power to the computer.
   c. Wait 10 seconds, then turn the power back on.

The system menus should appear on both monitors.

Connecting the mouse

The sample site data that has been installed on your system includes a mouse in the hardware configuration.

Connect the mouse to the COM2 port on the back of the computer unit.

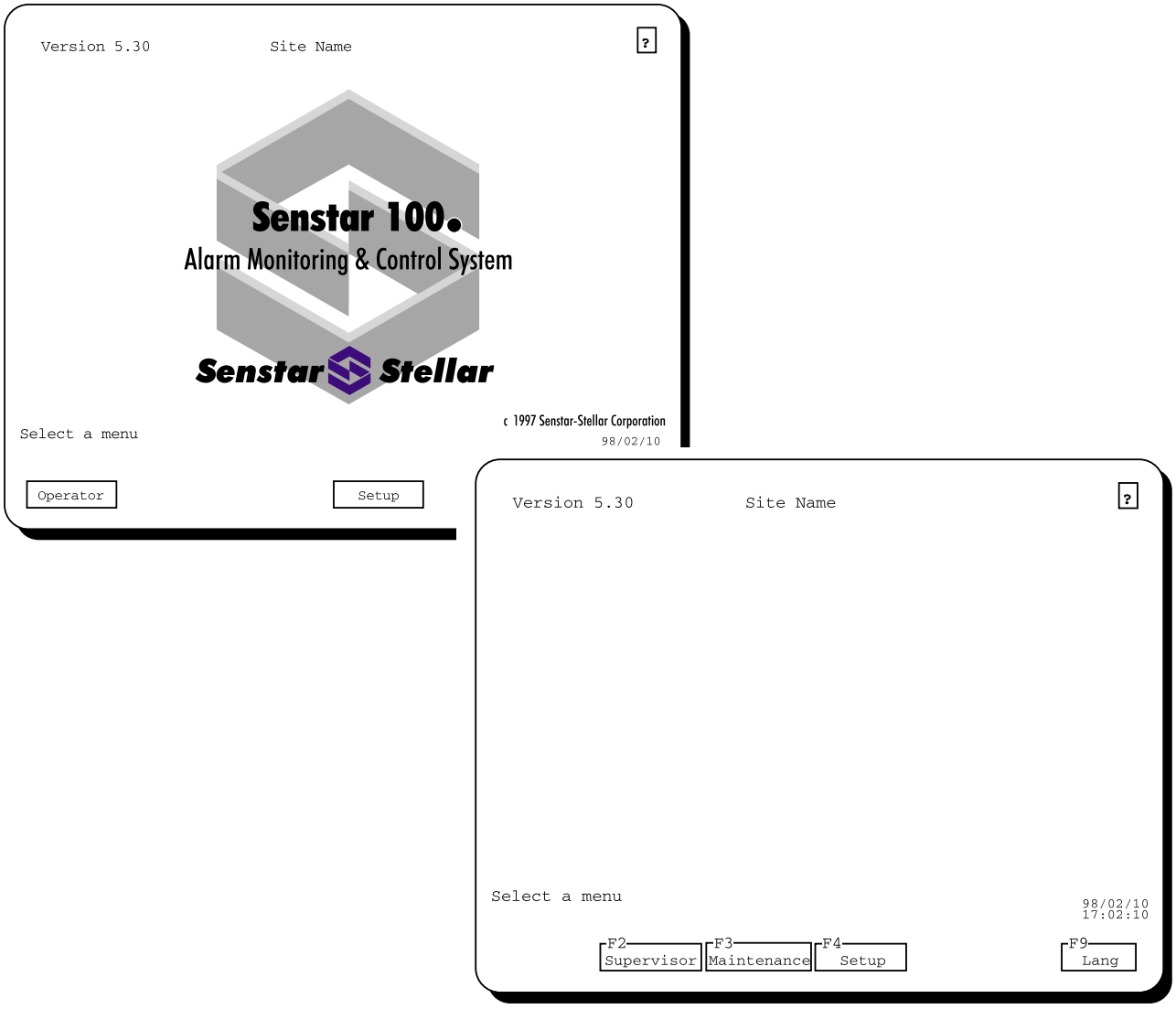
Loading the Sample Site Creation data (optional)

Follow this procedure if the Sample Site Creation data is not installed on your system AND you would like to view the sample database.

The Sample Site Creation data disk contains the source code for the sample site data that was installed on your system. To load the Sample Site Creation data disk you must:

- enter the Site Creation main menu
- create a new site database called 'Sample Site'
- load the Sample Site Creation data into the site database

If you've followed the procedure for installing the Site Creation software, you should see the system menus on the primary and secondary monitors.

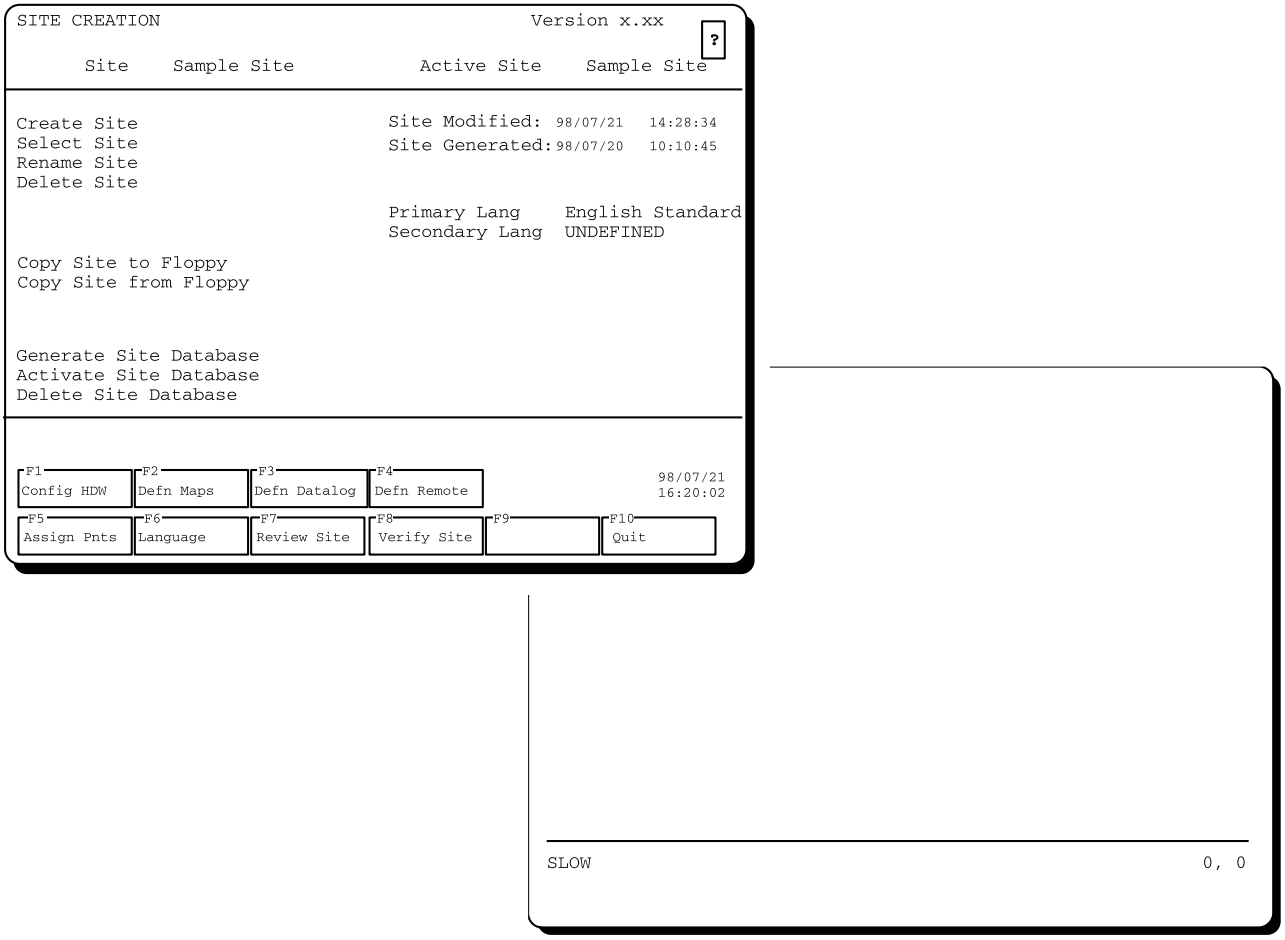


If the site data loaded for the application software does not match the hardware configuration of the computer, only the Maintenance and Setup menus will be available.

If the primary monitor displays a scrambled pattern, this indicates that the wrong monitor type has been selected in the site data. You can select either VGA or EGA. To change the monitor, reboot the system with the QNX boot disk and select F7 - Select monitor type, from the Startup menu.

To install the Sample Site Creation data:

1. Press the **F4 - Setup** function key to enter the Setup menu. The secondary screen changes to display the Setup menu. The primary screen doesn't change.
2. Press **F6 - Create Site** to enter the Site Creation software. You'll be warned that alarm processing capabilities will be disabled if you continue.
3. Press **F1** to continue. The Site Creation main menus appear on the monochrome and color monitors.



4. The **Create Site** function is highlighted. Press the **pick** key on the mouse or the **Ins** key on the keypad to select the function.
5. Key in the name **Sample Site**, and press **Enter**.

   The name appears in the **Site** field (top section of the monochrome screen) and the date and time appear in the **Site Modified** field.
6. Select the **Copy Site from Floppy** function. You're prompted to insert the Site Creation data disk.
7. Insert the Sample Site Creation data disk in the 'A' drive.
8. Press **pick** or **Ins** to start loading. You're prompted when loading is completed.

You now have a sample database, called 'Sample Site', on your system.

b System specifications

Senstar 100 specifications

Senstar 100 system

Senstar 100 is available in three sizes.

Sizes	Maps	Security devices
Small	4	2
Medium	16	4
Large	64	34

All systems include:

- an AT-type computer with 8 Mb RAM (depending on system size), 40 Mb (minimum) hard disk, floppy disk drive (3 1/2 in. or 5 1/4 in.), and an enhanced AT keyboard
- Senstar 100 application software and multifunction card
- a color monitor with optional touch screen
- optional secondary monitor (required for Site Creation application)
- user documentation

Hardware specifications

Primary devices

- Redundant control unit
- Video switcher (supports AD/RCA 1700, Broadcast Video, American Dynamics Matrix Switching System, Pelco 9500, Pelco 9750, Panasonic System 500, Burle Allegiant, Cohu MPC-M-104)
- Control Module
- Interface Unit
- Sennet
- Starcom devices
- custom devices
- DAVID 300
- Video controllers
- Fiber Sensys
- FOIDS

Secondary devices

- Event logger
- Report logger
- Mouse (supports Logitech C7 Microsoft Compatible, Logitech Mouseman, Logitech Trackman)
- Touch screen (supports Elographics Accu-Touch, Elographics Intellitouch, Smart Set)
- Modem

Site and map specifications

Site:	- 8 sites per system
	- 15 characters max. for site name
Map:	- 4, 16 or 64 maps per site
	- 15 characters max. for map name
Group:	- 16 groups per map
	- 15 characters max. for group name
Zone:	- 1 to 64 zones per map
	- 3 characters max. for zone label
Sensor:	- 1 to 3 sensors per zone
	- 3 characters max. for sensor label
Camera:	- 999 cameras per system/64 presets per camera/ 8 monitors
Datalog:	- 32 on/off states per site

Point assignment specifications

Alarms

- display
- diagnostic
- datalog
- remote
- duress

Display alarms

Sub-type of alarms	Input states	Output states
detection	alarm, acknowledge, reset, access, secure, entry/exit	alarm, unacknowledge, acknowledge, access, secure, current, timed access, entry delay, exit delay, input, select
tamper	alarm, acknowledge, reset, access, secure	alarm, unacknowledge, acknowledge, access, secure, current, timed access, entry delay, exit delay, input, select
jam	alarm, acknowledge, reset, access, secure	alarm, unacknowledge, acknowledge, access, secure, current, timed access, entry delay, exit delay, input, select
fail	alarm, acknowledge, reset access, secure	alarm, unacknowledge, acknowledge, access, secure, current, timed access, entry delay, exit delay, input, select

plus: camera - current

For each input state, define:
- device
- TM zone/card #
- point #
- type of point
- transition

For each output state, define:
- device
- TM zone/card #
- point

For cameras, define:
- camera #
- preset #
- camera classification

Diagnostic alarms

Sub-type of alarms	Input states	Output states
diagnostic	system-defined	alarm, unacknowledge, acknowledge, release, secure, current

plus: camera - current

For each output state, define:
- device
- TM zone/card #
- point

For cameras, define:
- camera #
- preset #
- camera classification

Datalog alarms

Sub-type of alarms	Input states	Output states
datalog	alarm	alarm, secure

For each input state, define:
- device
- TM zone/card #
- point #
- type of point
- transition

For each output state, define:
- device
- TM zone/card #
- point

Remote alarms

Sub-type of alarms	Input states	Output states
detection	alarm, acknowledge, reset, access, secure	alarm, unacknowledge, acknowledge, access, secure
tamper	alarm, acknowledge, reset, access secure	alarm, unacknowledge, acknowledge, access, secure
jam	alarm, acknowledge, reset, access, secure	alarm, unacknowledge, acknowledge, access, secure
fail	alarm, acknowledge, reset, access secure	alarm, unacknowledge, acknowledge, access, secure

For each input state, define:
- device
- TM zone/card #
- point #
- type of point
- transition

For each output state, define:
- device
- TM zone/card #
- point

Duress alarms

Sub-type of alarms	Input states	Output states
duress	acknowledge	alarm, secure, unacknowledge, acknowledge

For each input state, define:
- device
- zone/card #
- point
- alarm subtype
- transition

For each output state, define:
- device
- TM zone/card #
- point

Glossary

Access state change	a transition on the input point places the alarm into access.
Acknowledge state change	a transition on the input point acknowledges the alarm.
Acknowledge timeout	occurs when a user takes more than an assigned number of seconds to acknowledge an alarm. The number of seconds is assigned in the Setup menu (System Options function).
Active site	the site that is used by the system when you boot up. This is the site that will be seen at the Operator, Supervisor and Maintenance levels.
Alarm	a condition that occurs at your site. The five types of alarms are: display, diagnostic, datalog, remote and duress.
Alarm cause	feature that, if enabled, causes a list of alarm causes to appear when the operator is processing the alarm. The operator must select a cause from the list to finish processing the alarm.
Alarm state change	a transition on the input point generates an alarm.
Alarm sub-type	each alarm can be divided into sub-types. These are: sensor, tamper, jam and fail for display and remote alarms; diagnostic for diagnostic alarms; datalog for datalog alarms; and duress for duress alarms.
Camera assignment report	a report that lists the assignments for each camera.
Camera symbol	a white arrow that you place on your site map to indicate a camera location.
Class	the primary or secondary definition given to a map.
Color-graphics monitor	the monitor that you use to draw and display your site maps.
Command	on the monochrome screen, a command performs a particular function (e.g., copying, saving, deleting).
Communications parameters	baud rate, parity, data bits, stop bits and timeout. Parameters must be specified for each device in the system.

Console tampering occurs when a user makes three unsuccessful attempts to use a password (any menu).

Current-selection area the area of the monochrome screen where the menu title appears, as well as any applicable site, map or device names. You can't select or change any of the fields in this section.

Current state state in which the alarm is being viewed by the operator (i.e., in alarm processing, access/secure and video-stepping).

Data-entry area the area of the monochrome screen where you fill in data relating to a particular site, map or device, whose name is displayed in the current-selection area of the menu. This is also the area where you execute general commands for copying, saving or deleting data, and specific commands for drawing maps.

Datalog alarm an alarm that is printed on the event logger when the associated device changes state. It is identified by type, location and on/off state.

Datalog input assignment report a report that lists the input point assignments for any datalog alarms you've created.

Datalog output assignment report a report that lists the output point assignments for any datalog alarms you've created.

Datalog type a general way of grouping datalog alarms. It can be used for items of equipment (e.g., VCRs), or for occurrences (e.g., Guard Tour).

Device configuration report a report that lists the card or zone assignments for a selected primary device, and the inputs and outputs that can be used for each.

Device input assignment report a report that lists the input point assignments for a selected device.

Device label label that is automatically assigned to each primary device in the system (e.g., Control Module 1 = CM1).

Device output assignment report a report that lists the output point assignments for a selected device.

Device symbol a rectangular box that contains the three-character device label. You place this symbol on your site maps. The device symbol can appear on any map as long as you have a symbol for each primary device in your system. The box is white when you assign it to a map, green or light blue when in operation, and pink when in alarm.

Diagnostic — an alarm that is displayed on the site map when a hardware device or card fails. It is identified by the device or card label.

Diagnostic cameras report — a report that lists the point assignments for each camera assigned to a diagnostic alarm.

Diagnostic output assignment report — a report that lists the output point assignments for the diagnostic alarms on a selected map.

Diagnostic text — text that appears during diagnostics. Diagnostic text consists of: primary device names and labels, support device names, diagnostic locations (identifying messages, e.g., Card1), and diagnostic states (status messages, e.g., Fail, Fault, Off).

Display alarm — an alarm that is displayed on the site map when a sensor is activated. It is identified by map name, group, zone and sensor.

Display cameras report — a report that lists the point assignments for each camera assigned to a display alarm.

Display input assignment report — a report that lists the input point assignments for display alarms on a selected map.

Display output assignment report — a report that lists the output point assignments for display alarms on a selected map.

Done — the button on the mouse and the key on the keypad that terminates a current function.

Duress alarm — an alarm that is annunciated at a remote location when an operator enters a duress code at the control unit. It is identified by the name of the control unit (either the main unit or the redundant unit), and by the type of duress alarm.

Duress output assignment report — a report that lists the output point assignments for any duress alarm you've specified in your system.

English Standard — the default language of the system. English Standard can't be modified.

Enhanced alarm processing — option that lets you configure a display or remote alarm as a combination of several input points (maximum 8), and define alarm causes (see also JDAP, Alarm cause).

Entry delay — specified time period between the occurrence of an alarm and the annunciation of the alarm.

Error report in the Verify Site menu, a report that shows all errors and warnings related to your site.

Event logger the printer used to record operator actions and system events.

Event logger text text that is printed on the event logger. Event logger text consists of: event log text (portions of text that appear on the event logger under the location column), and event log states (status messages that print out on the event logger, under the status column).

Exit delay specified time period between the securing of an alarm (either by the operator or at the end of a timed access period) and the resetting of the alarm.

Function area the area of the monochrome screen where function keys are located, and where prompts, error messages and the time/date display appear.

Graphics section the area of the color-graphics screen where you draw your maps and where the maps are displayed.

Group a number of zones classed together based on a common element.

Hardware configuration the specifications for each device in your system. These include communications parameters, "normal" state of input and output lines, and test devices (if any).

Help text descriptions or explanations that appear on the help screens.

Input field a field on the monochrome screen where you enter a new name or value via the keyboard.

Input point a physical point on a hardware device, that collects data from the associated alarm. When activated, the input point causes the alarm to change state.

JDAP Joint Domain Alarm Processing. Feature in the Enhanced alarm processing option that lets you configure a display or remote alarm as a combination of several input points (maximum 8). (see also Enhanced alarm processing).

Location a way of identifying the datalog or remote alarm. The location can be a physical location, although this is not necessary.

Maintenance duress a duress alarm that occurs when a user accesses the Maintenance menu with a password that has the last digit replaced by the duress code.

Map background objects on your map such as buildings, rooms, fences and trees, that identify or depict your site.

Map grid an optional grid that you can place over your map to assist in accurate sizing and positioning of objects.

Map priority a number from 1 to 64 given to a map which sets its priority in relation to other maps. If alarms occur on more than one map at approximately the same time, the higher-priority map will have its alarms presented for processing first.

Map window the area of the color-graphics screen (639 pixels by 399 pixels) where you draw your site maps.

Monochrome monitor the monitor that you use to display menus, enter text information and perform most of the commands and functions.

Multifunction card card supplied by Senstar-Stellar, that contains software option GALs, sanity timer reset hardware and two serial ports.

Normal state the non-alarm state. It is defined by determining whether the data-collecting device uses On or Off to indicate the normal state. Depending on how the hardware is configured, a normal state could be normally open or normally closed; therefore, the database specifies On or Off, rather than Open or Closed.

On/off state status messages for datalog alarms that appear on the printer when the datalog alarm is activated or deactivated.

Operator access flag that allows you to make sensors inaccessible to the operator. If a sensor is inaccessible, the operator is not able to access the sensor during normal operator processing. The operator is able to access the sensor during alarm processing.

Operator duress a duress alarm that occurs when a user accesses the Operator menu with a password that has the last digit replaced by the duress code.

Output point a physical point on a hardware device that responds to data coming in from the associated alarm. It is activated based on the state that the alarm is in.

Password a six-digit code that you might need to type in before you can access the Setup menu.

Pick the button on the mouse and the key on the keypad that is used to select a highlighted function or field.

Point assignment — input and output points configured for each alarm in your system (display, diagnostic, datalog, remote and duress).

Pop-up menu — on the monochrome screen, displays a list of pre-defined or user-defined names or values from which you select what you want.

Port assignment — a Site Creation feature that takes all the information you've entered on cards, ports and hardware devices, and automatically assigns ports to each device.

Port configuration report — a report that lists all the ports in your system, the ports being used and the devices attached to them, and the communications parameters for each device.

Preset — camera view. Each camera can have up to 64 presets.

Primary device — a device that collects or responds to alarms. Primary devices include: control unit, redundant control unit, video switcher, Control Module, Interface Unit, Starcom devices and custom devices.

Primary language — the default language used by the system. The primary language is the only language used in the top level and logon menus and in automatically generated reports.

Primary map — when a sensor appears on more than one map, you can specify one map as the primary map. When an alarm is generated for the sensor, the primary map is displayed on the color screen.

Prompt line — displays information about the current system status, the function selected and operator instructions.

Remote alarm — an alarm that is annunciated at a remote location when a sensor is activated. It is identified by type and location.

Remote input assignment report — a report that lists the input point assignments for any remote alarms you've created.

Remote output assignment report — a report that lists the output point assignments for any remote alarms you've created.

Remote type — a general way of grouping remote alarms, for example, by the type of alarm that will be activated (Door alarm).

Report logger — the printer used to print maintenance and system reports.

Report text language that appears on reports. Report text consists of: report titles (report titles and column headings), report text (text that appears in the body of the report), and report labels (text that identifies a report when it is sent to the printer).

Reset state change a transition on the input point resets the alarm.

Sample site a demonstration site supplied by Senstar-Stellar, which you modify to indicate that you have a mouse; this enables you to use your mouse when creating your own site.

Screen text language that appears on the menu screens.

Secondary alarm group if an alarm in this group is not processed (accessed, secured or released) within a specified time, it is annunciated at a remote location.

Secondary device a device that assists in the running of your Senstar 100 system. Secondary devices include: printers, mouse, touch screen and modem.

Secondary language a second language used by the system to provide bilingual support.

Secondary map when a sensor appears on more than one map, you can specify one map as the primary map and the others as secondary maps. When an alarm is generated for the sensor, it is presented on the primary map.

Secure state change a transition on the input point places the alarm in a secure state.

Sensor a detection device used to protect your site.

Sensor label a one- to three-character identifier given to each type of sensor (e.g., door sensor = DOR, microwave sensor = MW).

Sensor priority a number from 0 to 64 given to a sensor which sets its priority in relation to other sensors on a map. If two sensors on a map generate an alarm at approximately the same time, the higher-priority one is presented for processing first.

Sensor symbol a box, triangle or star that encloses the sensor label.

Setup duress a duress alarm that occurs when a user accesses the Setup menu with a password that has the last digit replaced by the duress code.

Site the area that you want the Senstar 100 system to monitor.

Site Creation data — data supplied by Senstar-Stellar, that you need to install in order to modify your sample data.

Site data — information you must collect about your site, including all primary and secondary devices, site maps, and point assignments for all alarms in your system.

Site database — the database that you create with the Site Creation software. It holds all your site data.

Site error — in the Verify Site menu, a fatal condition that prevents your system from operating.

Site warning — in the Verify Site menu, a non-fatal condition that indicates that your system is not fully operational, even though it will run.

Starcom — Senstar-Stellar's standard communications protocol.

Status field — on the monochrome screen, displays a name or value that has been defined by another field on the menu, from another menu, or by the system.

Status line — the area of the color-graphics screen that displays the cursor speed (fast or slow), and the position of your cursor on the screen.

Style form — template for assigning output points. Style forms are practical if you use the same point information for alarms of the same type. One style form can be used for all alarms of the same type.

Supervisor duress — a duress alarm that occurs when a user accesses the Supervisor menu by using a password that has the last digit replaced by the duress code.

Timed access — specified time period that an alarm point remains in the access state.

Timeout — refers to the length of time (in seconds) after communication ceases, before a communication fail is detected.

Transition — the change from a 'normal' (non-alarm) state to an alarm state, or vice versa.

Undo — the button on the mouse and the key on the keypad that is used to cancel a function or point.

Zone — a division on the site map which represents an area covered by one to three sensors.

Zone label a one- to three-character identifier (usually numeric, but can include letters), given to each zone on the map.

Zone symbol a line or polygon that represents the area of detection covered by one to three sensors.

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

O

P

R

S

T

U

V

W

Z